ČERVENKA CONSULTING

Cervenka Consulting Ltd.
Na Hrebenkach 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

ATENA Program Documentation
Part 4-6

Tutorial for ATENA-GiD
3D Shear Beam

Step by step guide for nonlinear analysis
with ATENA and GiD

Written by

Zdenka Procházková, Jan Červenka, Zdeněk Janda, Dobromil Pryl

Prague, August 20, 2009

Trademarks:
ATENA is registered trademark of Vladimir Cervenka.
GiD is registered trademark of CIMNE of Barcelona, Spain.
Microsoft and Microsoft Windows are registered trademarks of Microsoft Corporation.
Other names may be trademarks of their respective owners.

Copyright © 2000-2009 Cervenka Consulting Ltd.

TABLE OF CONTENTS

1. INTRODUCTION 1

2. STARTING PROGRAM 2

3. PRE-PROCESSING 3

3.1 Introduction 3
3.1.1 Introduction of the graphical user interface 4

3.2 Geometrical model 8
3.2.1 Concrete beam 9
3.2.2 Loading and supporting steel plates 16
3.2.3 Reinforcement bars 28
3.2.4 Layers 31

3.3 Material parameters 41
3.3.1 Concrete beam 41
3.3.2 Loading and supporting steel plates 50
3.3.3 Reinforcement bars 55

3.4 Boundary conditions 63
3.4.1 Support 64
3.4.2 Displacement 70
3.4.3 Symmetry condition 78
3.4.4 Monitors 81

3.5 Intervals – Loading history 89

3.6 Mesh generation 91
3.6.1 Structured mesh 94

4. FE NON-LINEAR ANALYSIS 102

4.1 Missing contacts 104
4.1.1 Master Top beam condition 105
4.1.2 Slave Top plate condition 108
4.1.3 Master bottom beam and Slave bottom plate conditions 110

4.2 AtenaWin interface description 112

4.3 Load-displacement graph 114

4.4 Crack width display 119

5. POST-PROCESSING 127

5.1 GiD post-processing 127

5.2 AtenaWin post-processing 134

6. CONCLUSION 137

7. PROGRAM DISTRIBUTORS AND DEVELOPERS 138

8. LITERATURE 140

1. INTRODUCTION

This tutorial provides a basic introduction to the usage of the program **ATENA** and **GiD,** and it is specifically targeted for **ATENA-GiD** beginners. **ATENA-GiD** is a finite element based software system specifically developed for nonlinear analysis of reinforced concrete structures. **ATENA** is used for the analysis itself and the program **GiD** is used for data preparation and mesh generation.

This tutorial contains a step by step explanation how to perform a non-linear analysis on an example problem of a reinforced beam without smeared reinforcement. The geometrical and material properties correspond to the experimental setup by Leonhard in 1962. More details about the problem or experiment can be also obtained from the original report [6] or from the program developer or distributor.

It is possible to create and analyse the example problem described in this tutorial in demo version of **ATENA-GiD**. Because of that a rather coarse finite element mesh is used.

The step by step demonstration is performed on an example of simply supported beam, which is loaded by two loads as it is shown in Figure 1. The problem is symmetric around its vertical axis; therefore, only one symmetric half of the beam will be analyzed.

It is recommended to print-out this version, in order to easily follow the instructions. In case of printing, it is advisable to use both sided and colour printing.

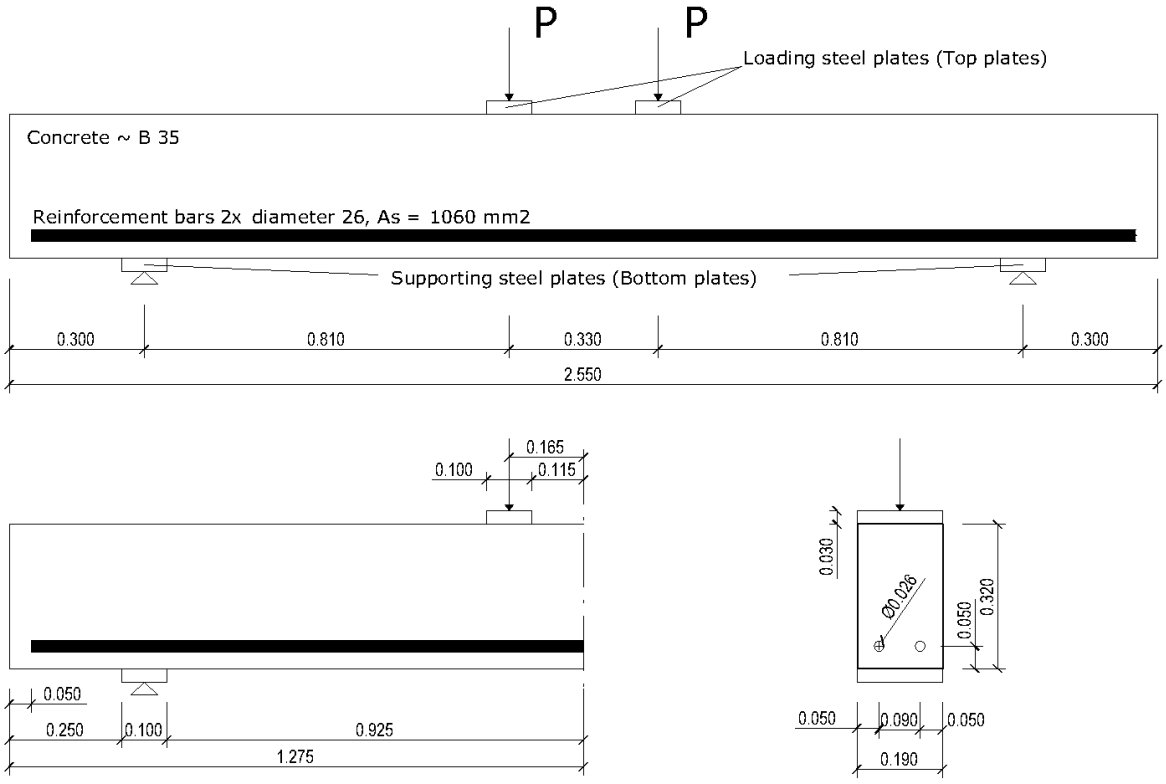


Figure 1: Geometry of the structure

The steps necessary for the data preparation, non-linear analysis and post-processing are depicted on subsequent figures, which show the computer screen for each step and the corresponding user action. There is always also a short description for each figure.

It should be noted that not all features of **ATENA-GiD** system are described in this manual. For more details about the data preparation and post-processing, the user is encouraged to read the manual of the program **GiD** and **ATENA-GiD** manual [2].

2. STARTING PROGRAM

Before using the **ATENA-GiD** system it is necessary to install it on your computer. The programs **GiD** and **ATENA** can be installed using the standard **ATENA** installation. At the end of the installation the user must select the installation of **GiD** and **ATENA-GiD** interface. After that your computer should be ready to run the example problem described in this document. The installation process is described in detail in ATENA-GiD manual [2].

In order to start a nonlinear analysis in **ATENA-GiD** system, first the program **GiD** is started. The recommended version is 9.0.4 or newer (the oldest supported version is 7.7.2b). The program **GiD** can be started from the start menu of your computer using the following path: **`Start | All Programs | CervenkaConsulting | ATENA Science | GiD`**.

This opens the program **GiD**, which is used for the preparation of the numerical model of the analyzed structure. This process is described in the subsequent Chapter 3. The execution of the nonlinear analysis is described in Chapter 4 and the post-processing in Chapter 5.

3. PRE-PROCESSING

3.1 Introduction

This chapter explains the basic steps, which are to be performed in order to define a complete geometrical, and then a finite element model for non-linear FE analysis by **GiD**.

The purpose of the geometrical model is to describe the geometry of the structure, its material properties and boundary conditions. The analytical model for the finite element analysis will be created during the pre-processing with the help of the fully automated mesh generator.

The definition of the geometry starts with the creation of geometrical points. These points are later connected into boundary lines, and then surfaces are defined by selecting appropriate bounding lines. Volumes can be formed either by extrusion of surfaces or manually by selecting all bounding surfaces. Three-dimensional regions are modelled by volumes in **GiD**. The reinforcement is modelled as a line. These reinforcement lines are not usually connected to any surface or volume, but they usually lie inside the volumes entities that form the concrete structure.

After creation of the geometry, material properties should be defined and assigned to individual volumes. Boundary conditions are used to define supports and loads. The boundary conditions and loads are defined in **GiD** with the help of "Intervals". Interval represents a set of boundary conditions and loads that are applied in a specified number of steps. An appropriate definition of intervals can be used to specify a complete loading history.

In **ATENA** analysis it is always useful to define monitoring points. The monitoring points are used to see the evolution of certain quantities during the analysis. For instance they can be used to follow the development of deflection or forces at given locations.

3.1.1 Introduction of the graphical user interface

Before starting the definition of the geometrical model it is good to introduce the graphical user interface of **ATENA-GiD**. The main window is shown in the Figure 2. It shows the basic layout of **GiD** program right after its start and it explains the basic functionality of the various icons and menus.

This window shows the basic layout of the **GiD** program. At this stage it contains only commands for the creation of geometric objects. In order to activate **ATENA** specific materials and boundary conditions, an appropriate problem type needs to be selected. This is described in the next section.

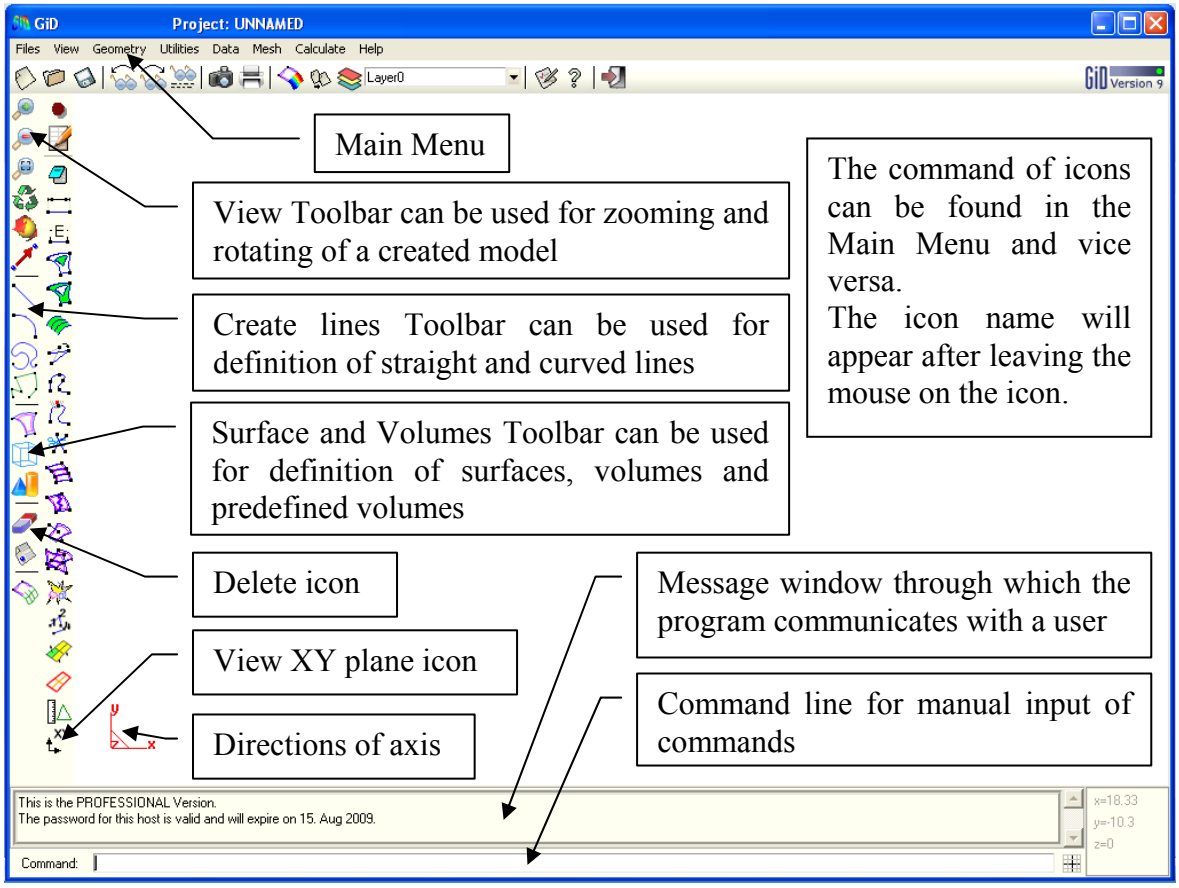

GiD
Project: UNNAMED
Files View Geometry Utilities Data Mesh Calculate Help
Layer0
GiD Version 9
Main Menu
View Toolbar can be used for zooming and rotating of a created model
Create lines Toolbar can be used for definition of straight and curved lines
Surface and Volumes Toolbar can be used for definition of surfaces, volumes and predefined volumes
Delete icon
View XY plane icon
Directions of axis
The command of icons can be found in the Main Menu and vice versa.
The icon name will appear after leaving the mouse on the icon.
Message window through which the program communicates with a user
Command line for manual input of commands
This is the PROFESSIONAL Version.
The password for this host is valid and will expire on 15. Aug 2009.
Command:

Figure 2: Graphical user interface of ATENA-GiD.

3.1.1.1 Problem type

The **GiD** is a general-purpose pre- and post-processing tool for variety of numerical problems (and analysis software). The **GiD** can be customized to create input data for basically any finite element software. The customization is done through the definition of various problem types. Each problem type represents certain customization. Therefore it is important to select an appropriate problem type at the beginning of the work.

In this case **ATENA** problem type has to be selected. The problem type definition must be done before starting input of data. Executing this command later may cause losing of all material and load definition. The problem type is selected from the Main menu **`Data | Problem Type | AtenaV4 | Static`**. Once this is selected **ATENA** specific icons will appear in the main window (see Figure 3).

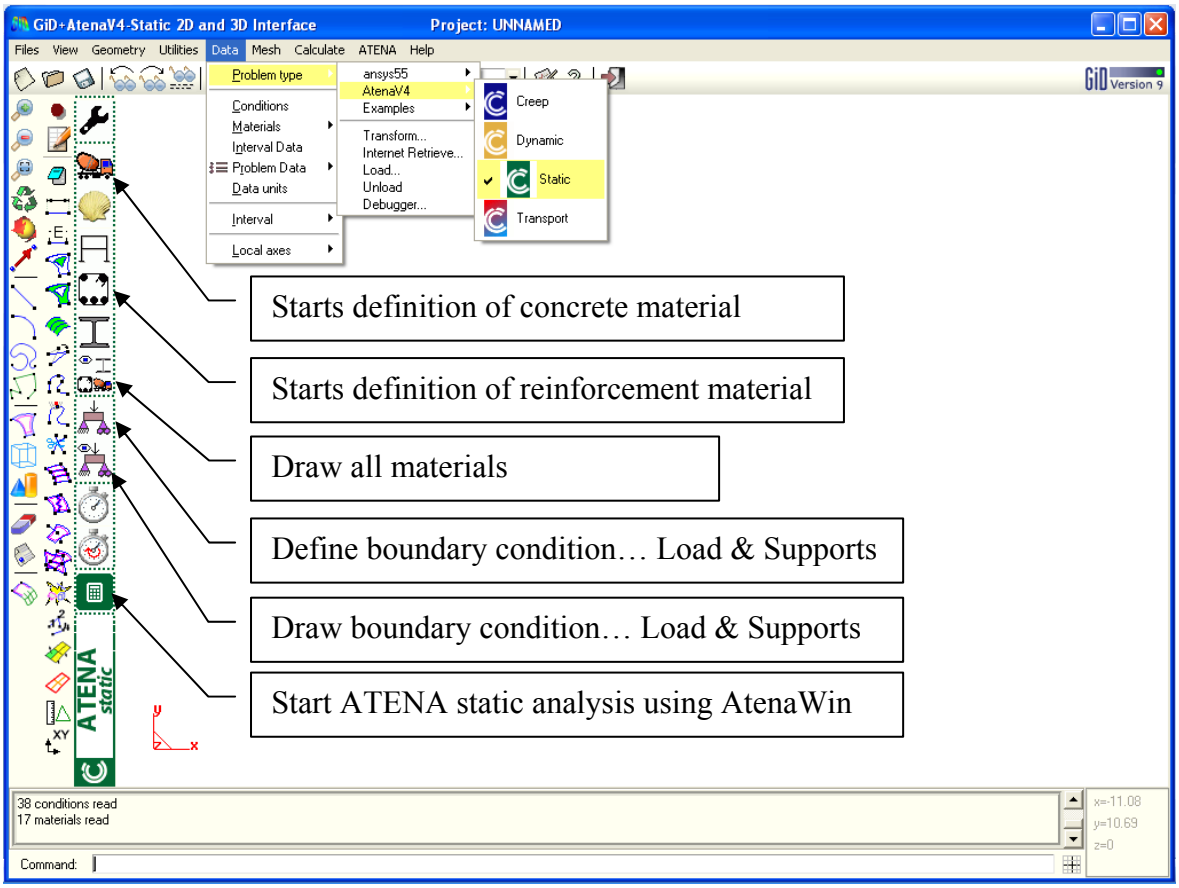


Figure 3: Problem type menu and basic ATENA icons.

It is also recommended to explore the help of the program **GiD**. This can be found in the Main menu or by pressing F1 on your keyboard.

It is also much recommended to save file and also regularly save created model during the formation of the geometrical model. Saving is done by selecting **`File | Save`**. The name of the document can be chose for example **`3DBeam`**.

3.1.1.2 Problem data

When the analysis is finished, all results are saved in files. From those files results can be executed and processed lately. Therefore it is useful to rename the title of the files where results should be saved and it is useful to do this saving in the beginning of the any creation of project. Later it could be forgotten.

It is done by the command **`Data | Problem Data | Problem Data`** in the main menu (see Figure 4).

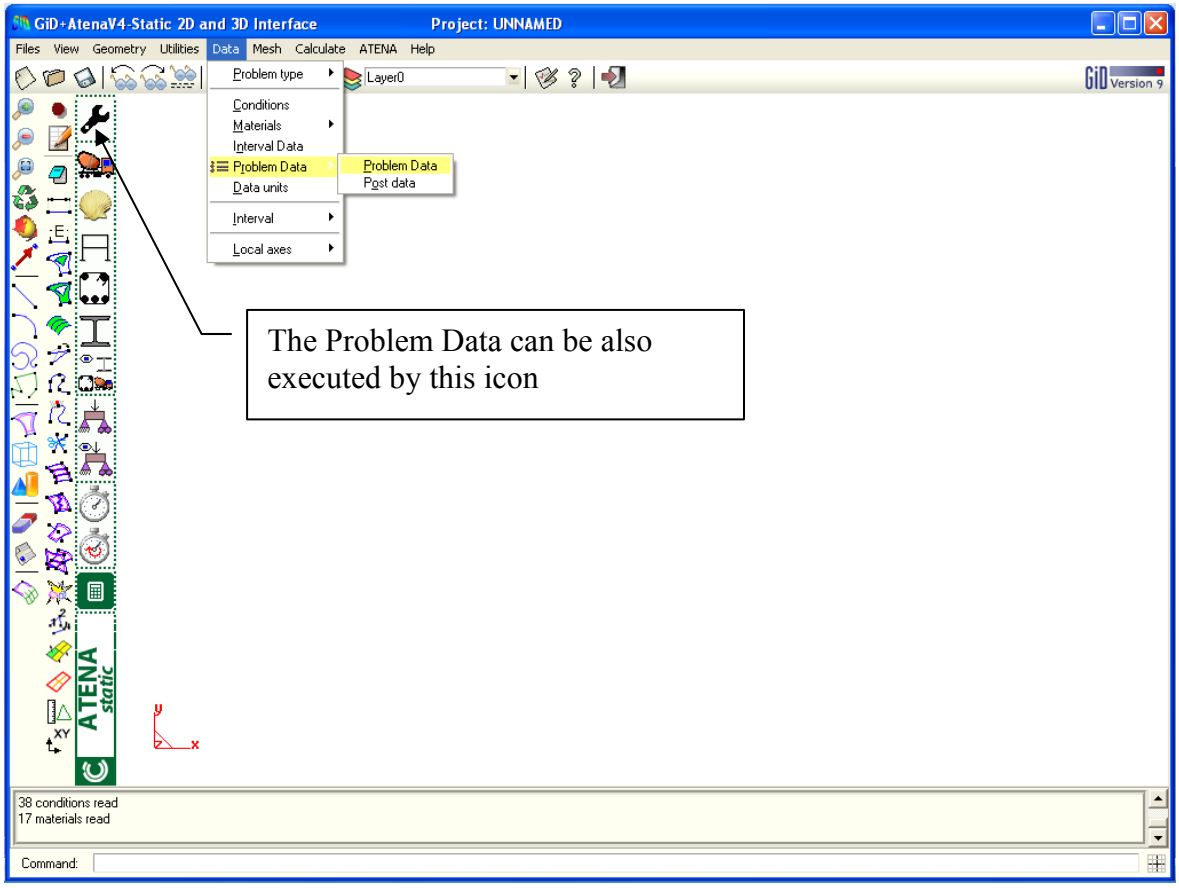


Figure 4: The command Problem Data

After selecting this command the Problem Data window will appear (see Figure 5). There the Title and Task Name should be changed to rename files where the results of the analysis will be saved.

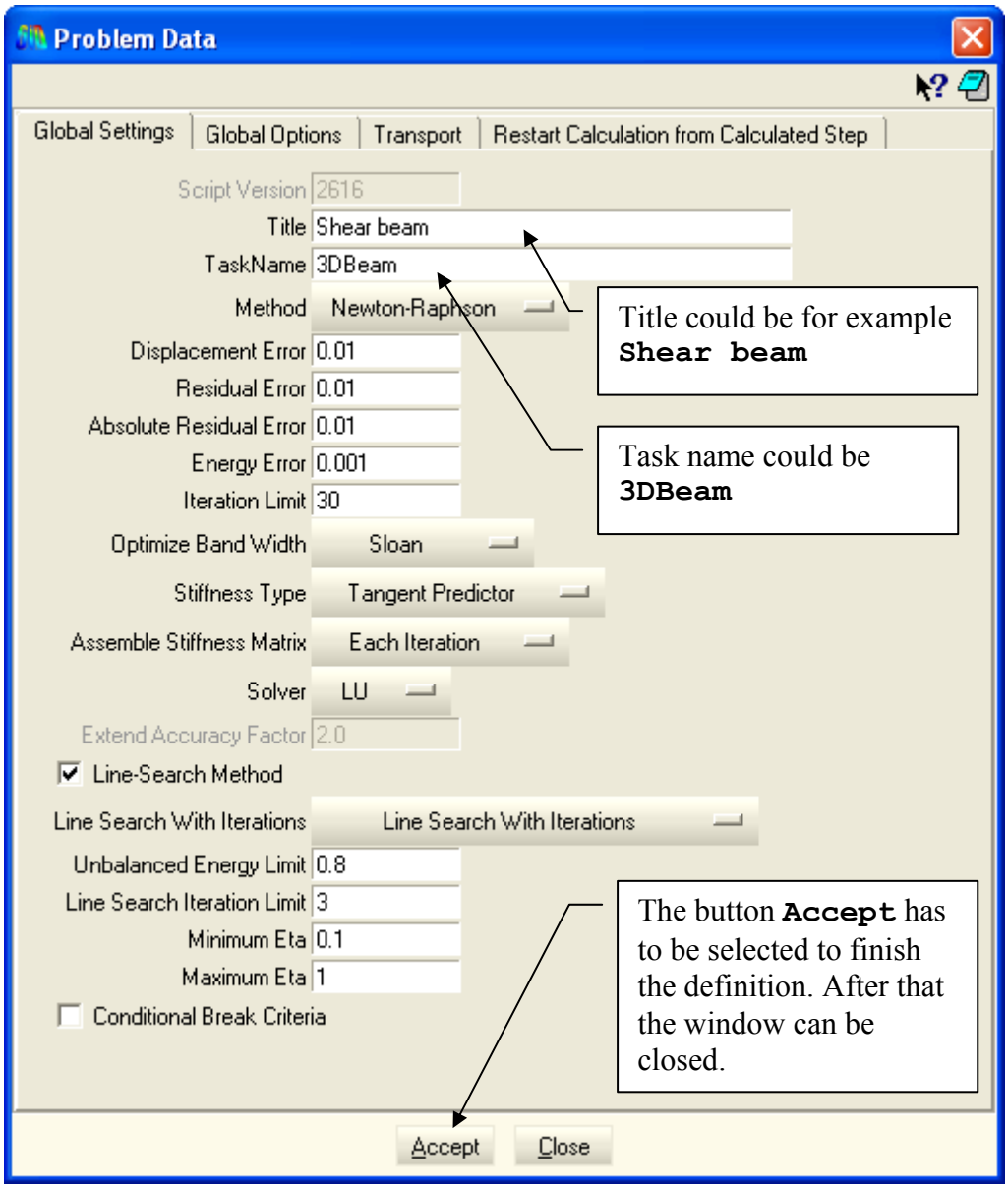


Figure 5: The Problem Data definition

Parameter input:

Title: Shear beam

TaskName: 3DBeam

3.2 Geometrical model

This chapter describes definition of the geometrical model. Because the beam is symmetric, only half of the beam will be created in this example.

The geometrical model of this half beam (see Figure 6) is composed of three 3D regions and reinforcement. In **GiD** the 3D regions are called "Volumes". Therefore the geometrical model contains three volumes – beam, loading and supporting plates. The reinforcement is modelled by two straight lines. The definition of these geometrical entities is described in the subsequent chapters.

It is useful to use the layer function for the definition of the geometrical model. It is a function, where particular parts of the model can be placed on different layers and then displayed, hidden or locked etc. In this geometrical model three separate layers will be created – beam layer, plates layer and reinforcement layer.

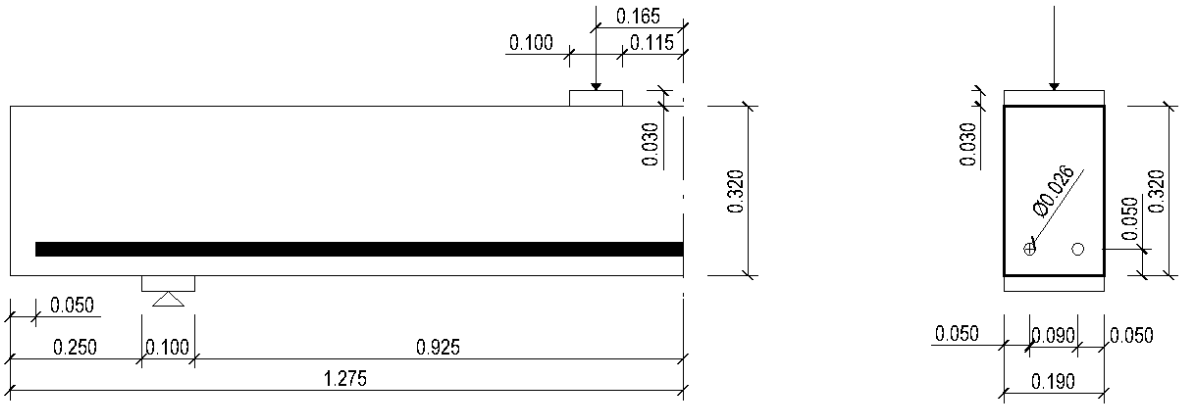


Figure 6: The geometrical model is composed from three volumes – beam and two plates

3.2.1 Concrete beam

A concrete beam forms the main part of the example. This section describes the definition of the three-dimensional beam geometry. The geometry of the beam will be created by an extrusion of a rectangular surface. That will be defined by four lines.

First step is to create points, which will be later connected into a rectangular surface. A point is created using the command **`Geometry | Create | Point`** in the Main menu. In order to create a rectangle, four points are needed. Each point is defined by three coordinates (x,y,z). The coordinates of points should be written in the command line in the bottom part of the main window.

The coordinates can be written all together separated by comma. A dot represents a decimal point. The definition of coordinates of each point is completed by ENTER. (In the command line it is very handy to use the key arrow up and down on your keyboard to view previously entered coordinates. These previous coordinates can be changed and entered again.)

In this case the following points should be entered:

Parameter input:
Coordinates of points:
1: (0,0,0)
2: (1.275,0,0)
3: (1.275,0.19,0)
4: (0,0.19,0)

NOTE: The table named "**Parameter input:**" will guide you through the whole tutorial. This table shows the parameters, which should be entered. There are predefined parameters in some dialogs windows for the definitions of parameters. Then the table **Parameter input:** shows only parameters, which should be changed.

After entering coordinates the points appear in the graphical area (see Figure 7). It is useful to enlarge the model such that it fills the whole screen. For that the command **`View | Zoom | Frame`** in the main menu or the Frame icon can be used (see Figure 8).

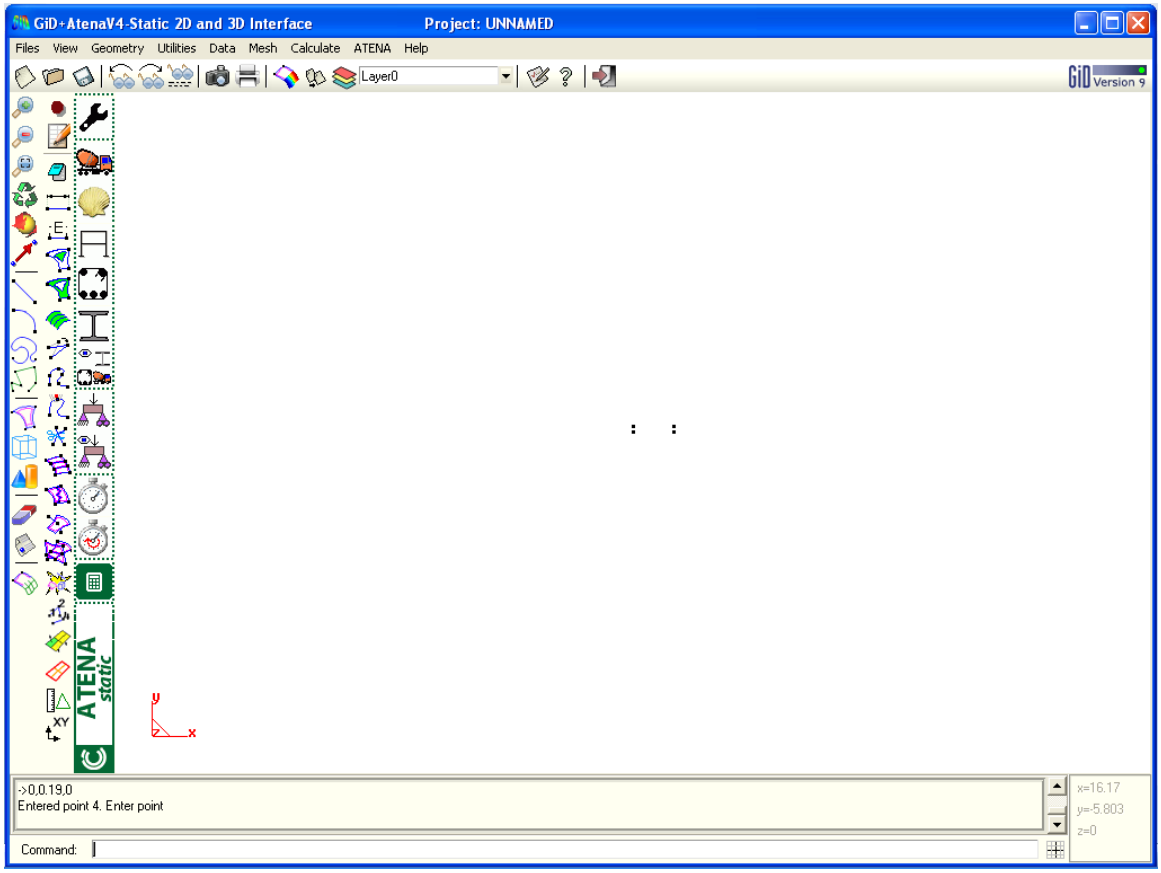


Figure 7: Four created points before zooming.

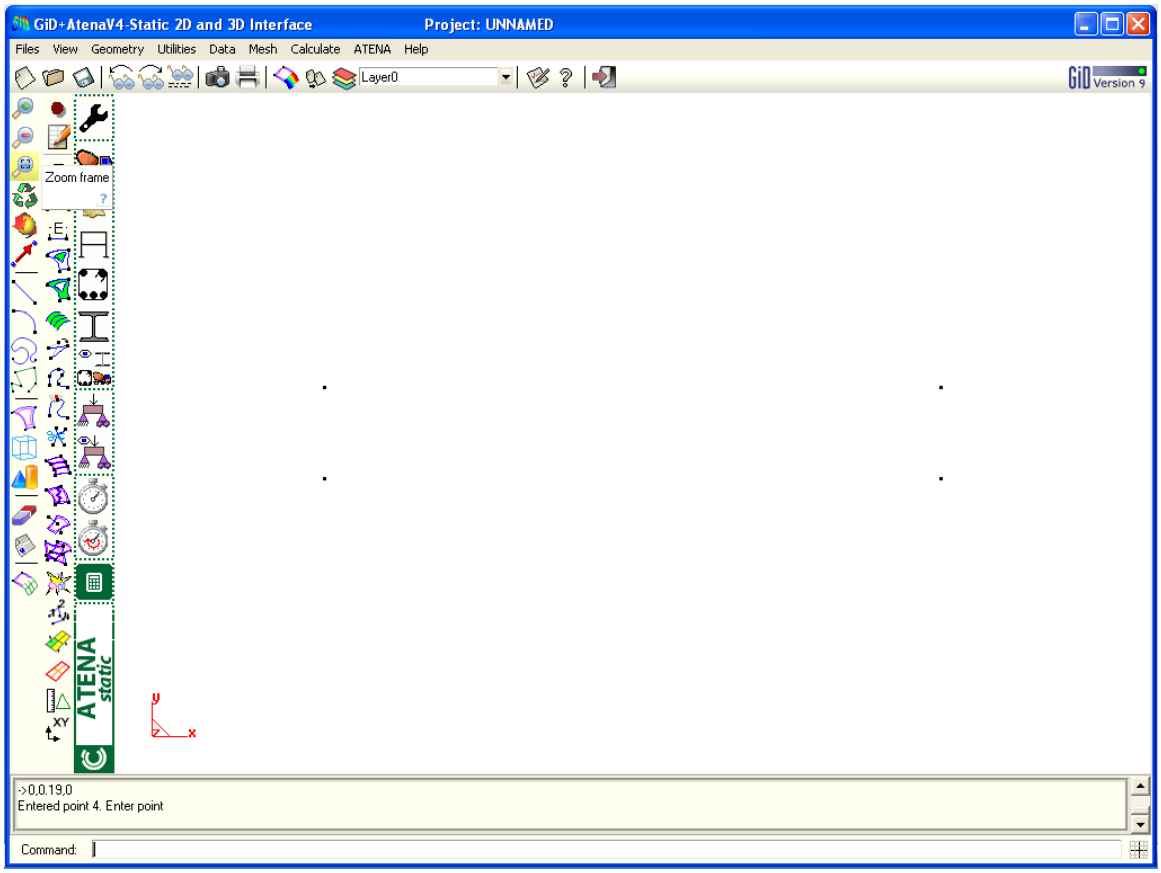


Figure 8: Using of the Zoom frame icon enables to have a better view of the created geometry.

Next step is to connect these points by lines. Lines are created using the command **`Geometry | Create | Straight line`** in the Main menu or by clicking on the icon . Then the message window at the bottom will show the sentence: *Enter points to define line (ESC to leave)*. The lines can be defined by entering exact coordinates into a command line or it is possible to directly pick the already existing points. In this example the direct picking has been chosen.

The direct picking can be done by selecting **`Contextual | Join Ctrl-a`** in the Mouse menu. The Mouse menu can be found by clicking on the right button of the mouse in the graphical area (see Figure 9).

Alternatively this option can be activated directly by pressing the key Ctrl and 'a' at the same time.

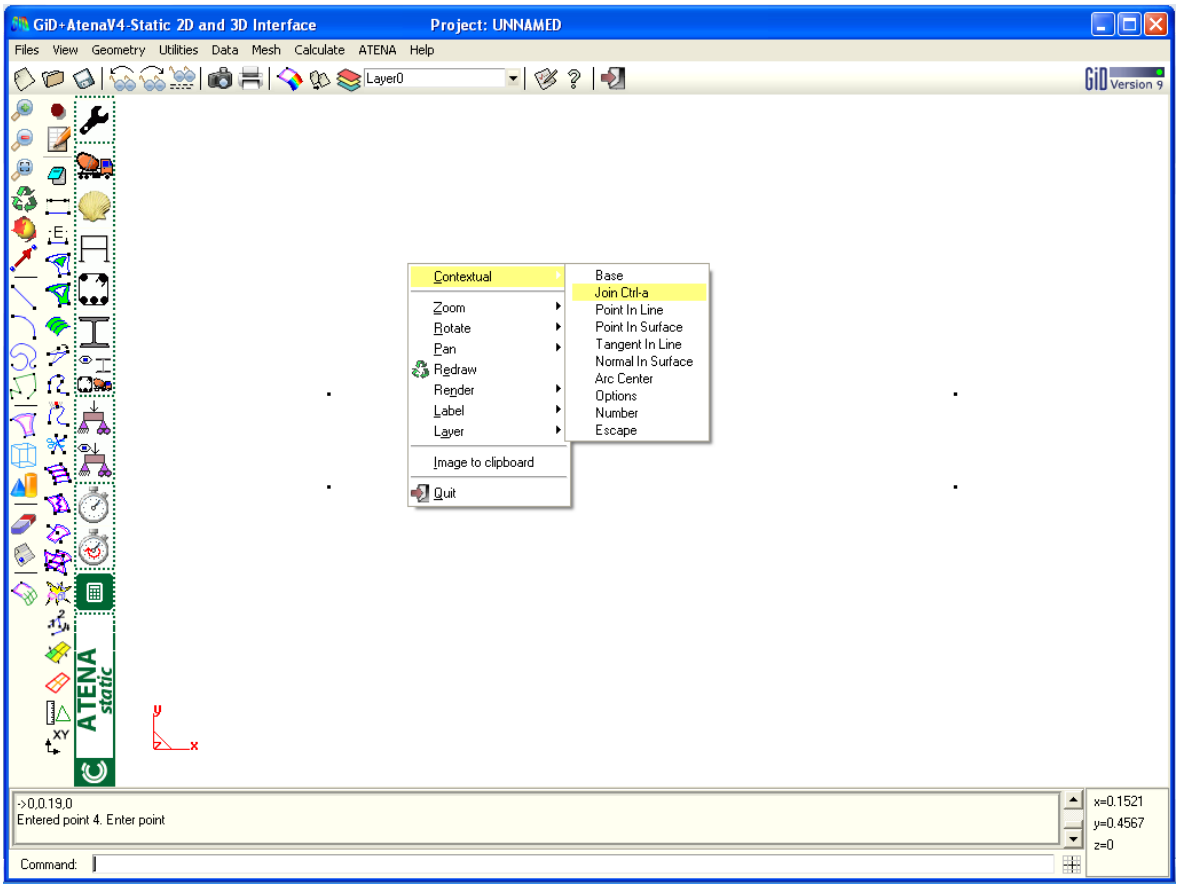


Figure 9: The Join function in the Mouse menu.

After selecting the join function the mouse cursor will change to this shape . Then after clicking into a graphical area the nearest point will be selected. Now all points can be connected by lines into the rectangle (see Figure 10). The create line function should be finished by pressing ESC key.

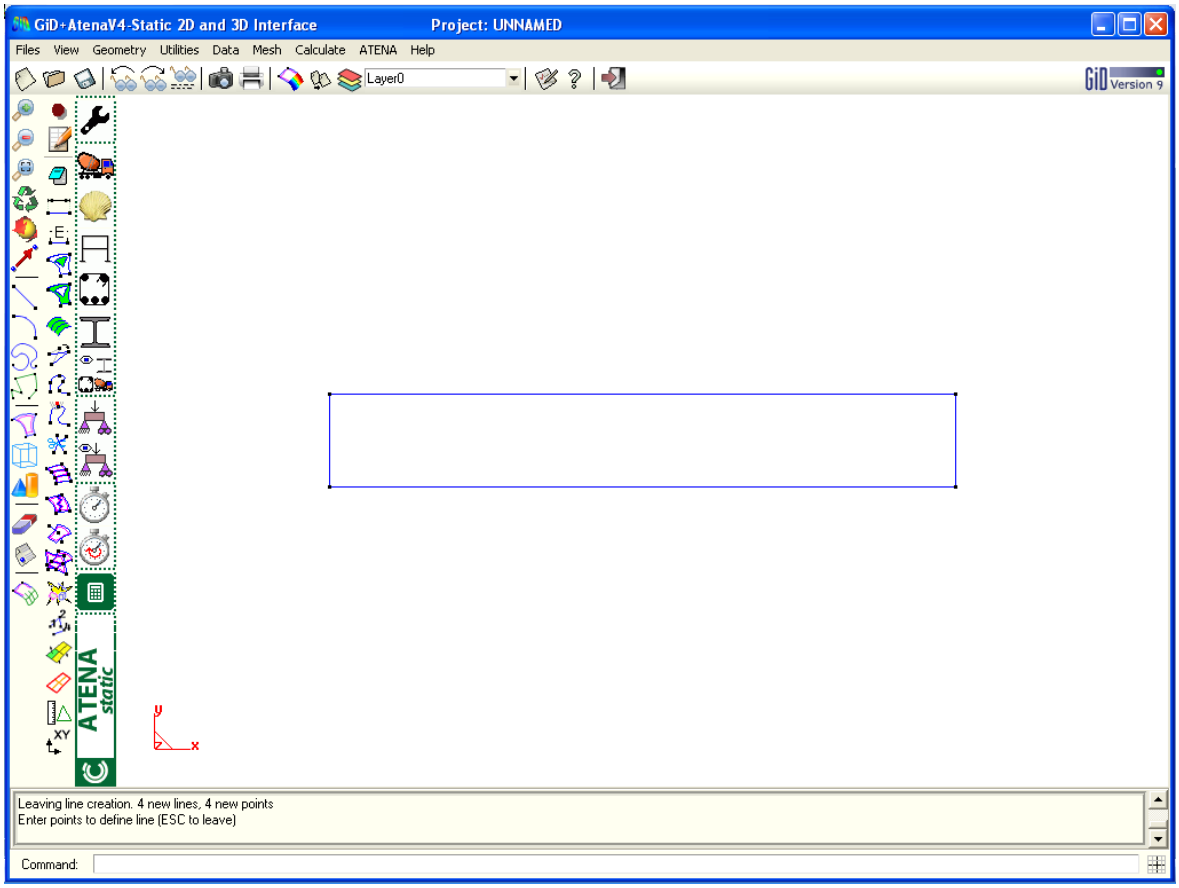

GiD+AtenaV4-Static 2D and 3D Interface
Project: UNNAMED
Files View Geometry Utilities Data Mesh Calculate ATENA Help
Layer0
GiD Version 9
ATENA static
Leaving line creation. 4 new lines, 4 new points
Enter points to define line (ESC to leave)
Command:

Figure 10: The lines connected into a rectangle.

The **GiD** distinguishes four types of entities – point, lines, surfaces and volumes. In our case there are already two entities - points and lines. Lines define a rectangular boundary but it is not a surface until a surface is defined. Therefore a surface has to be created.

It is done by selecting **`Geometry | Create | NURBS surface |By contour`** in the main menu and then selecting all lines defining the required surface in the graphical area (see Figure 11). Clicking on the icon can also start the **`Create surface`** function. Next, the lines bounding the surface should be selected, and then it is necessary to press ESC key to complete the surface definition. The newly created surfaces are denoted by a pink colour as seen in the Figure 11.

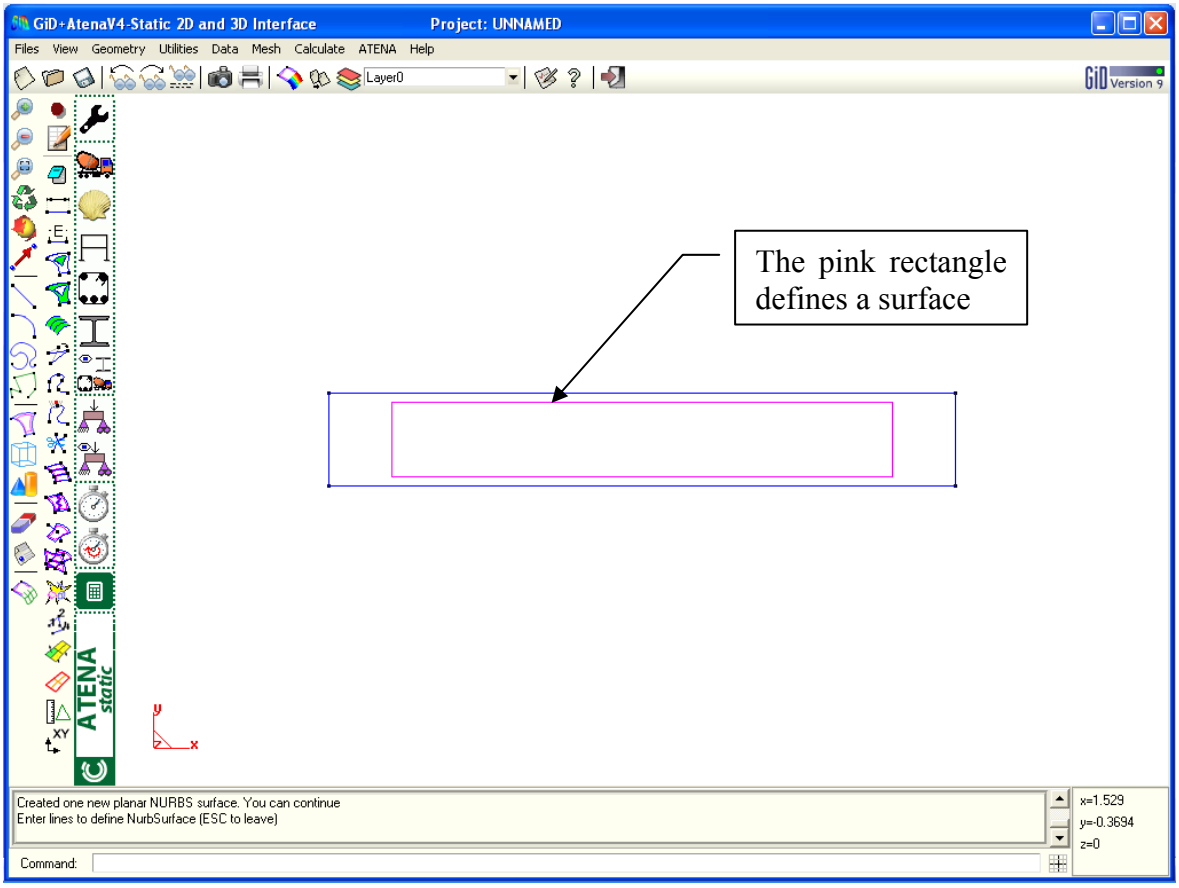


Figure 11: The pink rectangle in the middle of blue lines defines the added surface.

The next step is to extrude the created surface into a volume to obtain the required beam. The extrusion is done by the command **Copy**, which appears after selecting the command **Utilities | Copy** in the main menu (see Figure 12).

In this example, the surface is extruded in the direction of the Z-axis over the beam thickness **0.32 m**. The thickness will be given by the vector defined by coordinates of two points set in the **Copy** menu. The definition of the extrusion is depicted in the Figure 12.

After the definition of all parameters the **Select** button should be pressed. Then the surface required for the extrusion can be selected in the graphical area. After the selection of surface it is necessary to press **Finish** button to complete the extrusion.

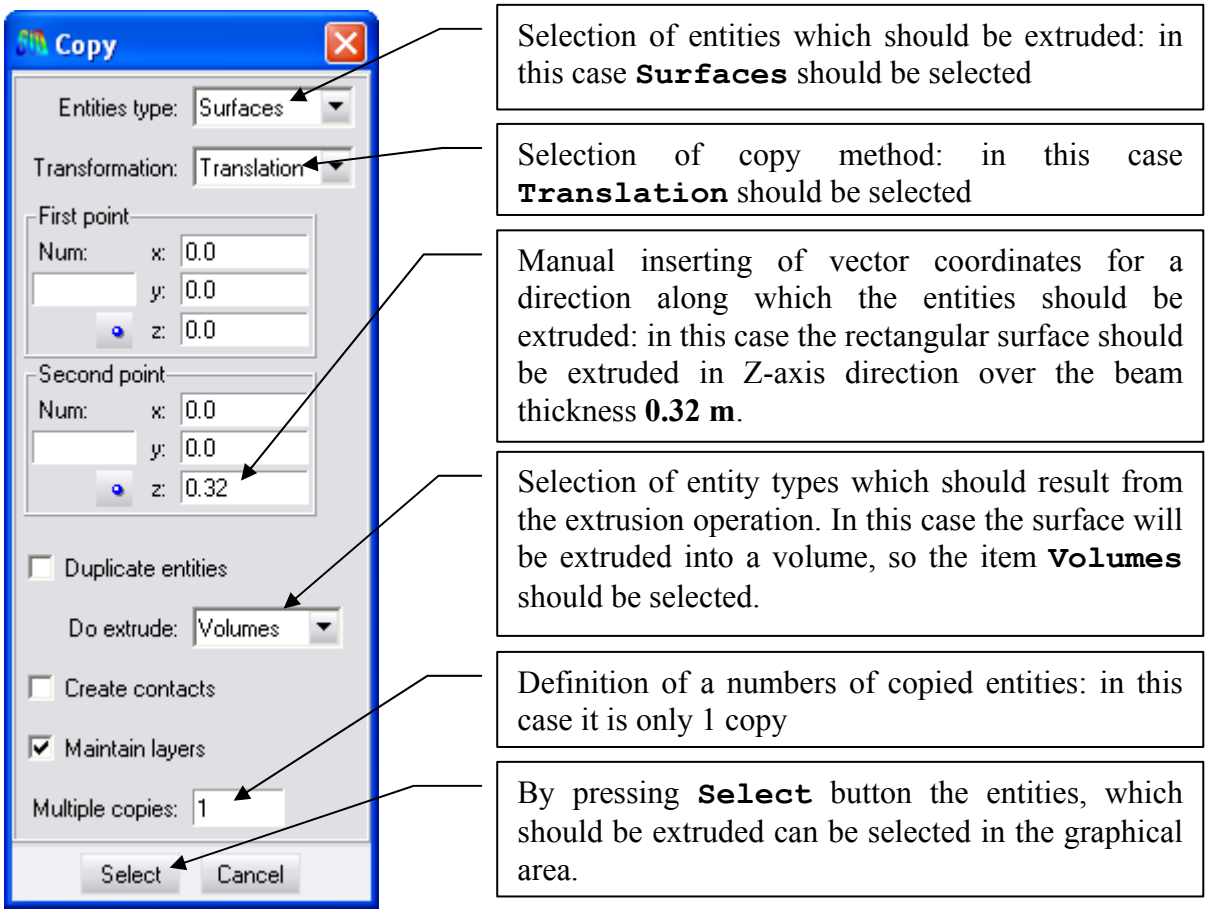

Copy
Entities type: Surfaces
Transformation: Translation
First point
Num:
x: 0.0
y: 0.0
z: 0.0
Second point
Num:
x: 0.0
y: 0.0
z: 0.32
Duplicate entities
Do extrude: Volumes
Create contacts
Maintain layers
Multiple copies: 1
Select
Cancel
Selection of entities which should be extruded: in this case Surfaces should be selected
Selection of copy method: in this case Translation should be selected
Manual inserting of vector coordinates for a direction along which the entities should be extruded: in this case the rectangular surface should be extruded in Z-axis direction over the beam thickness 0.32 m.
Selection of entity types which should result from the extrusion operation. In this case the surface will be extruded into a volume, so the item Volumes should be selected.
Definition of a numbers of copied entities: in this case it is only 1 copy
By pressing Select button the entities, which should be extruded can be selected in the graphical area.

Figure 12: The description of Copy menu

Parameter input:
Entities type: Surfaces
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.0
z: 0.32
Do extrude: Volumes

The selection of the surface can be done by a direct clicking on the pink line, which defines a surface. Another option is to select the surface by holding the right mouse button and by moving of the mouse. The box should cross at least one line of the surface to be selected. After the proper selection the pink selected surface will change to the red colour.

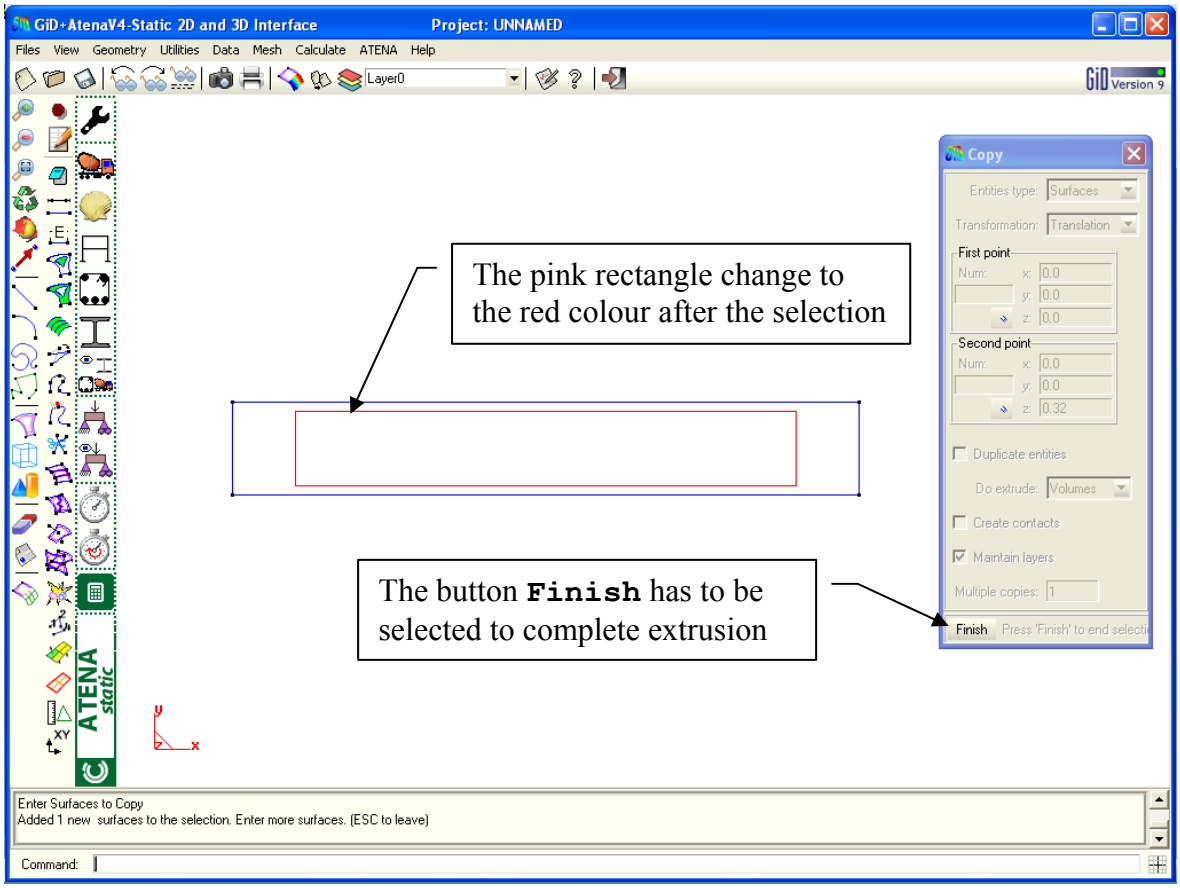

GiD+AtenaV4-Static 2D and 3D Interface
Project: UNNAMED
Files View Geometry Utilities Data Mesh Calculate ATENA Help
Layer0
GiD Version 9
Copy
Entities type: Surfaces
Transformation: Translation
First point
Num: x: 0.0 y: 0.0 z: 0.0
Second point
Num: x: 0.0 y: 0.0 z: 0.32
Duplicate entities
Do extrude: Volumes
Create contacts
Maintain layers
Multiple copies: 1
Finish Press 'Finish' to end selecti
The pink rectangle change to the red colour after the selection
The button Finish has to be selected to complete extrusion
ATENA static
Enter Surfaces to Copy
Added 1 new surfaces to the selection. Enter more surfaces. (ESC to leave)
Command:

Figure 13: The selection of the surface for the extrusion

To see the extruded volume it is possible to use Rotate Trackball icon or holding left mouse button + SHIFT key (see Figure 14).

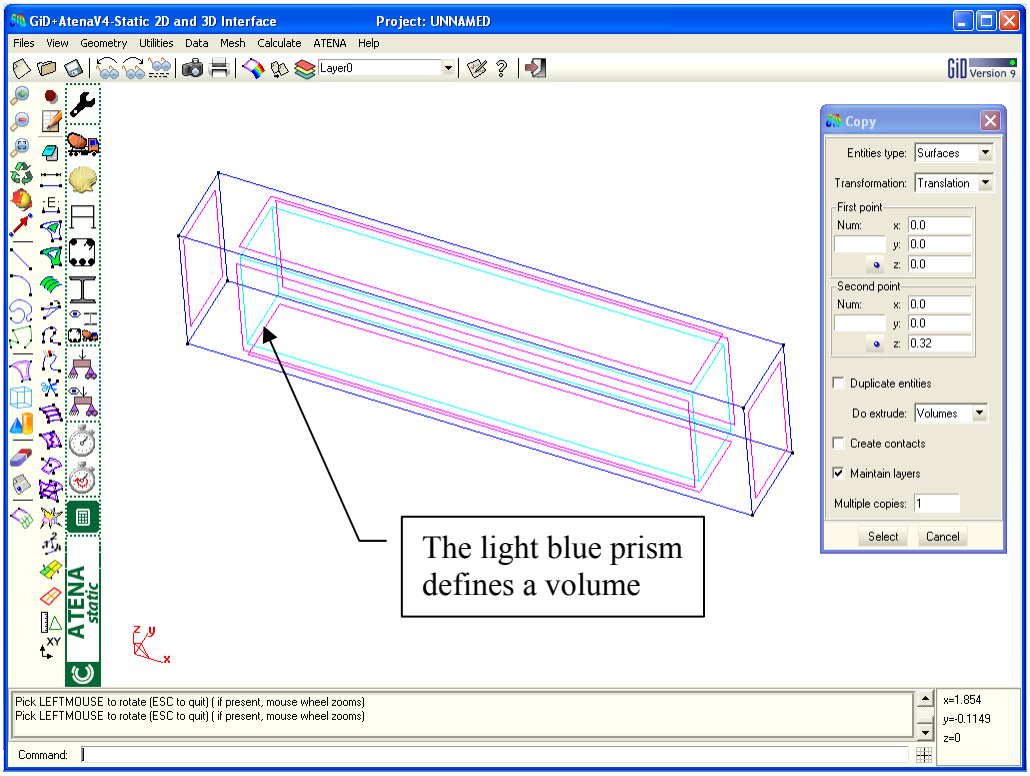


Figure 14: The extruded volume – the light blue prism defines a volume

3.2.2 Loading and supporting steel plates

After the creation of the beam geometry loading and supporting plates should be created. The top plate (loading plate) will be created first. The bottom plate (supporting plate) will be created by copying of the top plate.

The top plate will be created with using the commands **`Copy`** and **`Create lines`**. These commands should be known from the previous chapter. The dimensions and location of the plates can be seen on Figure 15.

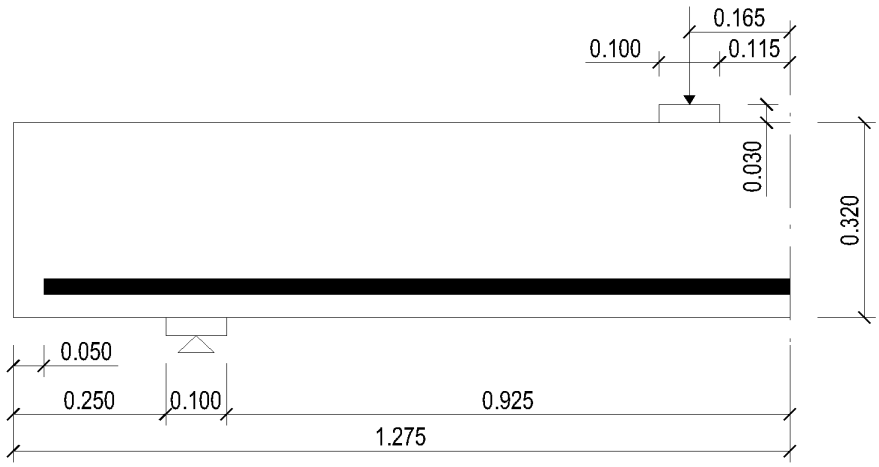


Figure 15: The dimensions of the half beam and location of steel plates

3.2.2.1 Top plate

It is useful and easier to use existing elements for the creation of a new object. The top plate is located on the right corner of the created beam. Therefore the upper-right edge of the beam can be copied and moved to **0.115 m** from the right end. Then this line will be copied and moved again. The second copy operation should move the line by a distance identical to the width of the steel plates. These two lines will be then connected into a rectangle. The surface will be added to this rectangle and then this surface will be extruded into a volume of the steel plate.

Before starting copying it is better to zoom in the right beam corner (see Figure 16). The **`Zoom in`** is activated by command **`View | Zoom | In`** or by clicking on the icon . The command **`Zoom in`** and **`out`** can be also activated by holding SHIFT key and using mouse scroll (In that case it is also necessary to move the view of the geometry. It can be done by holding SHIFT + right mouse button.).

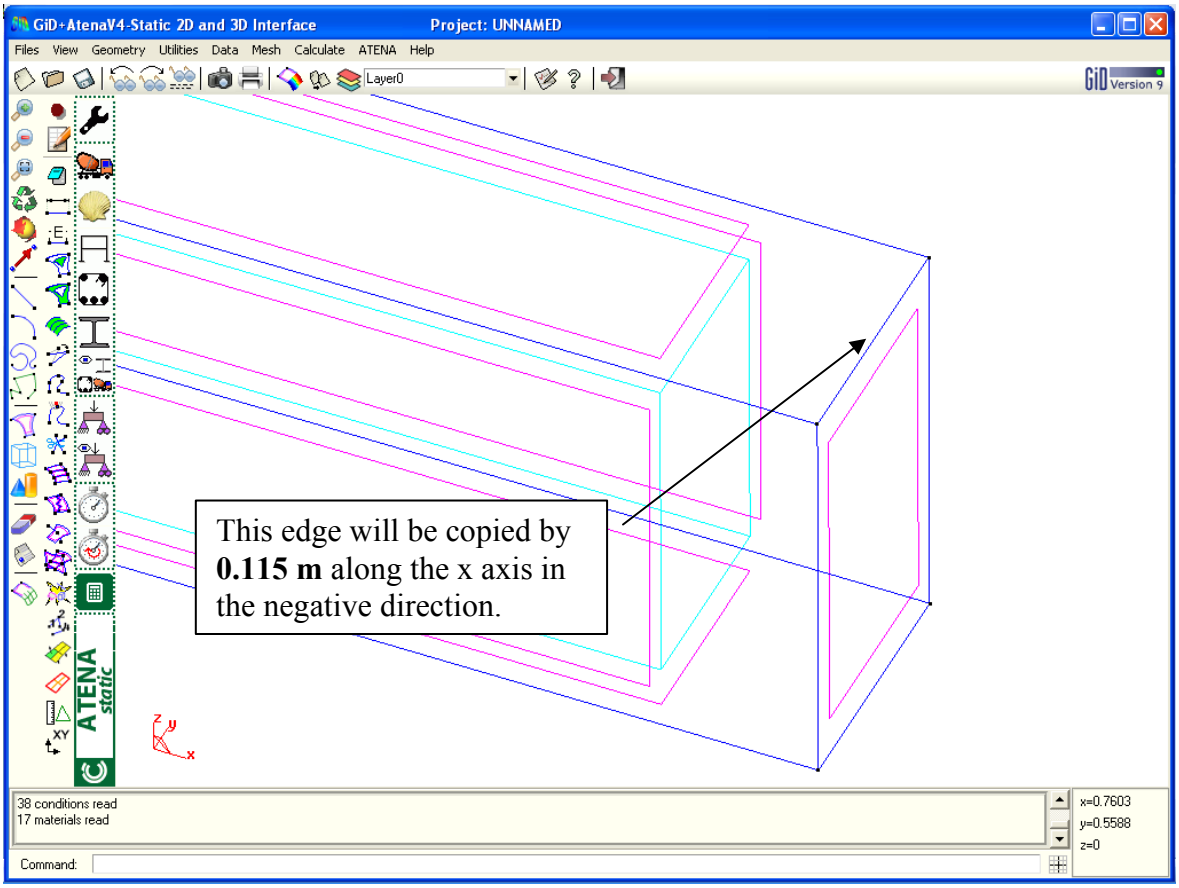
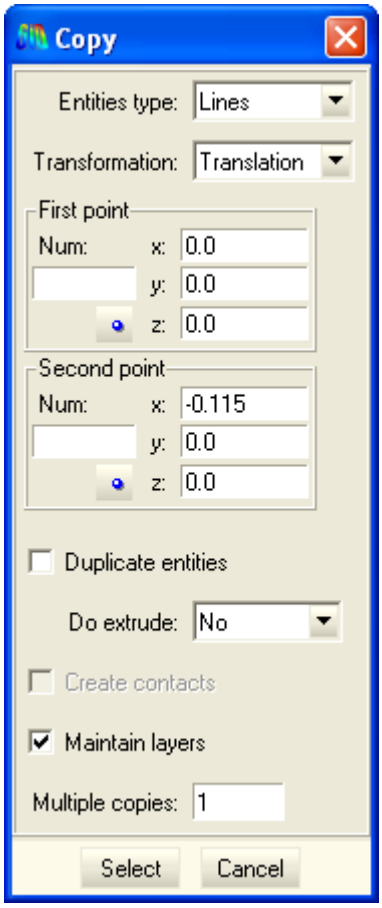


Figure 16: The geometry after Zoom in

The **Copy** menu appears after selecting **Utilities | Copy** in the Main menu. The new line should be in the **0.115 m** distance from the right edge of the beam. The copied entity is line, and there is no extrusion necessary. The parameter definition is depicted in the Figure 17. After the definition of all parameters, the **Select** button should be pressed. Then the line required for the copying can be selected in the graphical area (see Figure 18). After the selection of the line it is necessary to press **Finish** button to complete the translation (see Figure 19).



Parameter input:
Entities type: Lines
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: -0.115
y: 0.0
z: 0.0
Do extrude: No

Figure 17: The definition of translation of the line

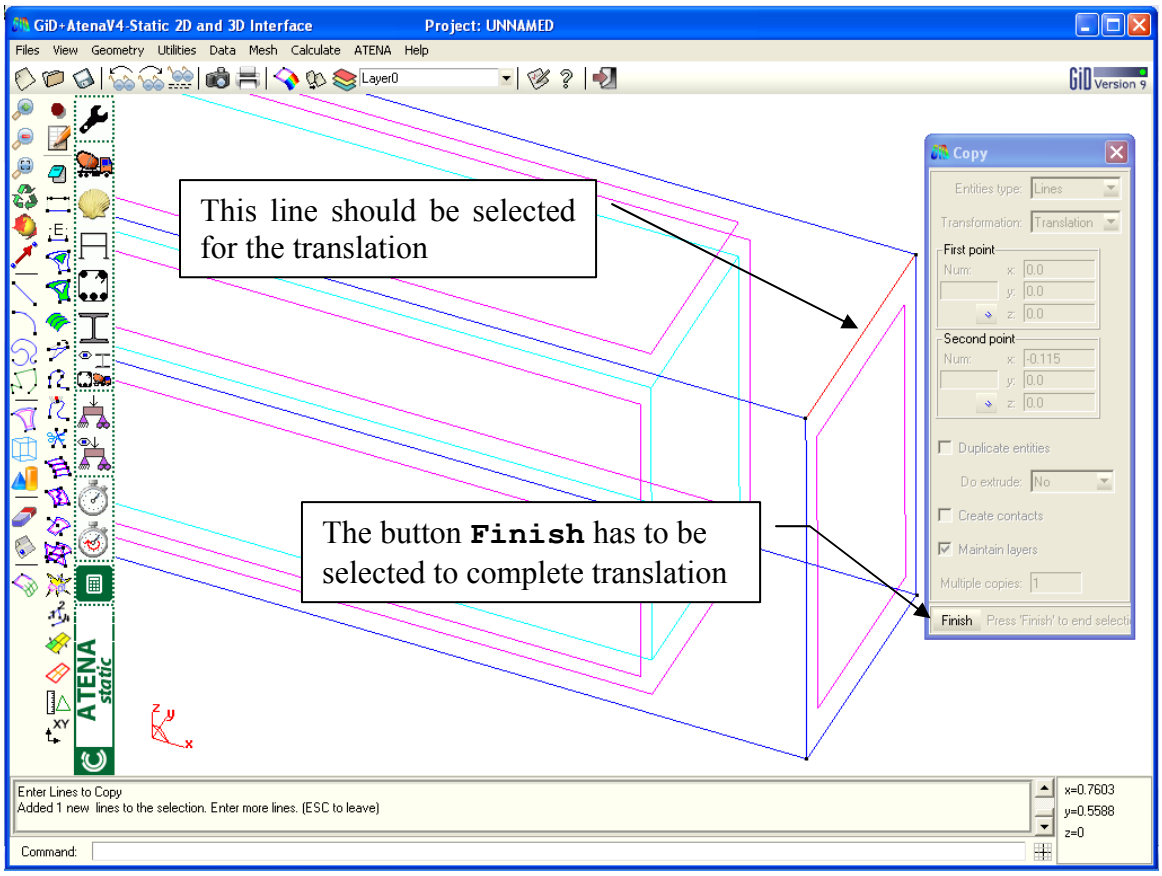


Figure 18: The selection of the line which should be copied

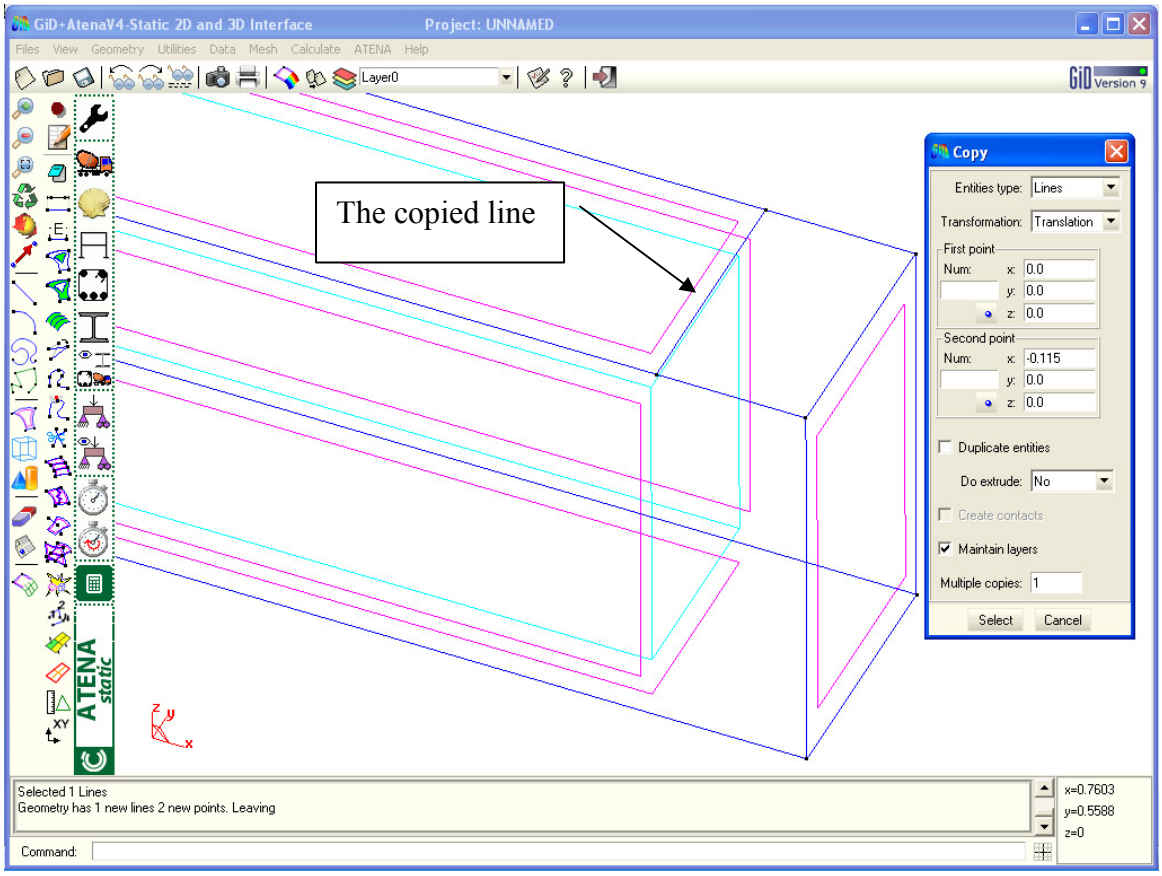
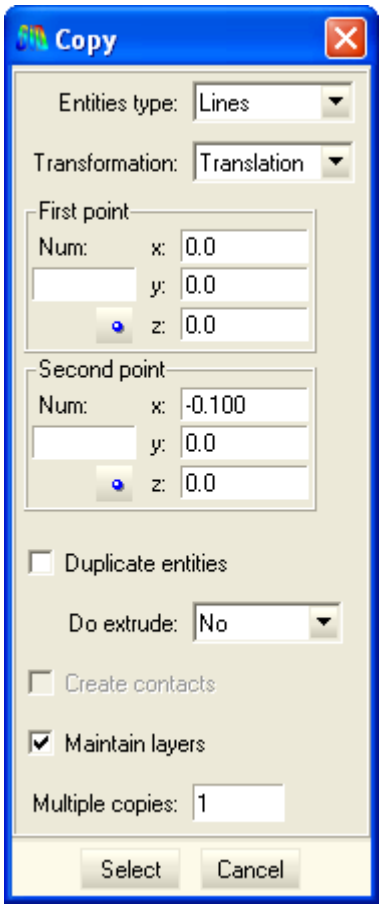


Figure 19: The new copied line

Now the new line will be copied again to create the second edge of the top plate. The width of the plate is 0.100 m. Therefore the second line will be translated by **0.100 m**. The parameter definition is depicted in the Figure 20. After the definition of all parameters the **`Select`** button should be pressed. Then the line required for copying can be selected in the graphical area (see Figure 21). After the selection of the line it is necessary to press **`Finish`** button to complete the translation (see Figure 22).



Parameter input:
Entities type: Lines
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: -0.100
y: 0.0
z: 0.0
Do extrude: No

Figure 20: The parameter definition of the second line

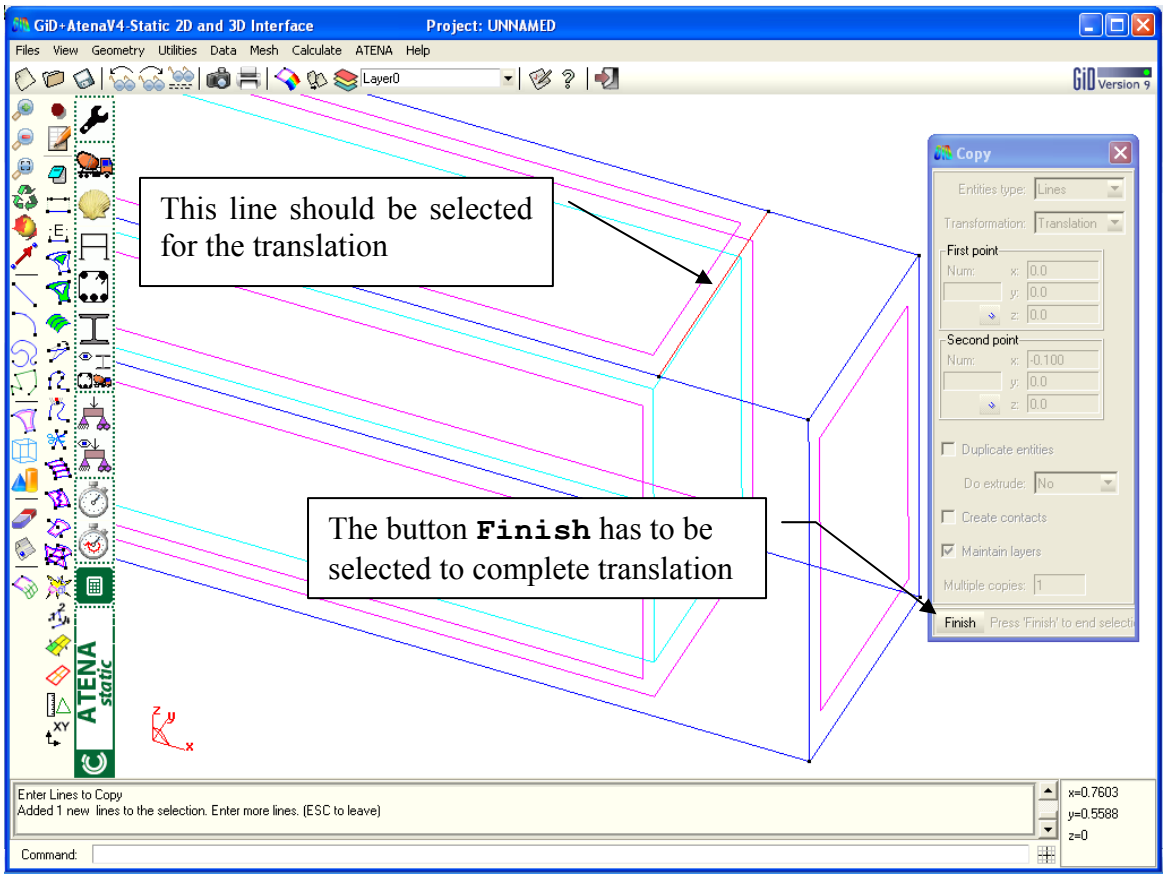


Figure 21: The selection of the line which should be copied

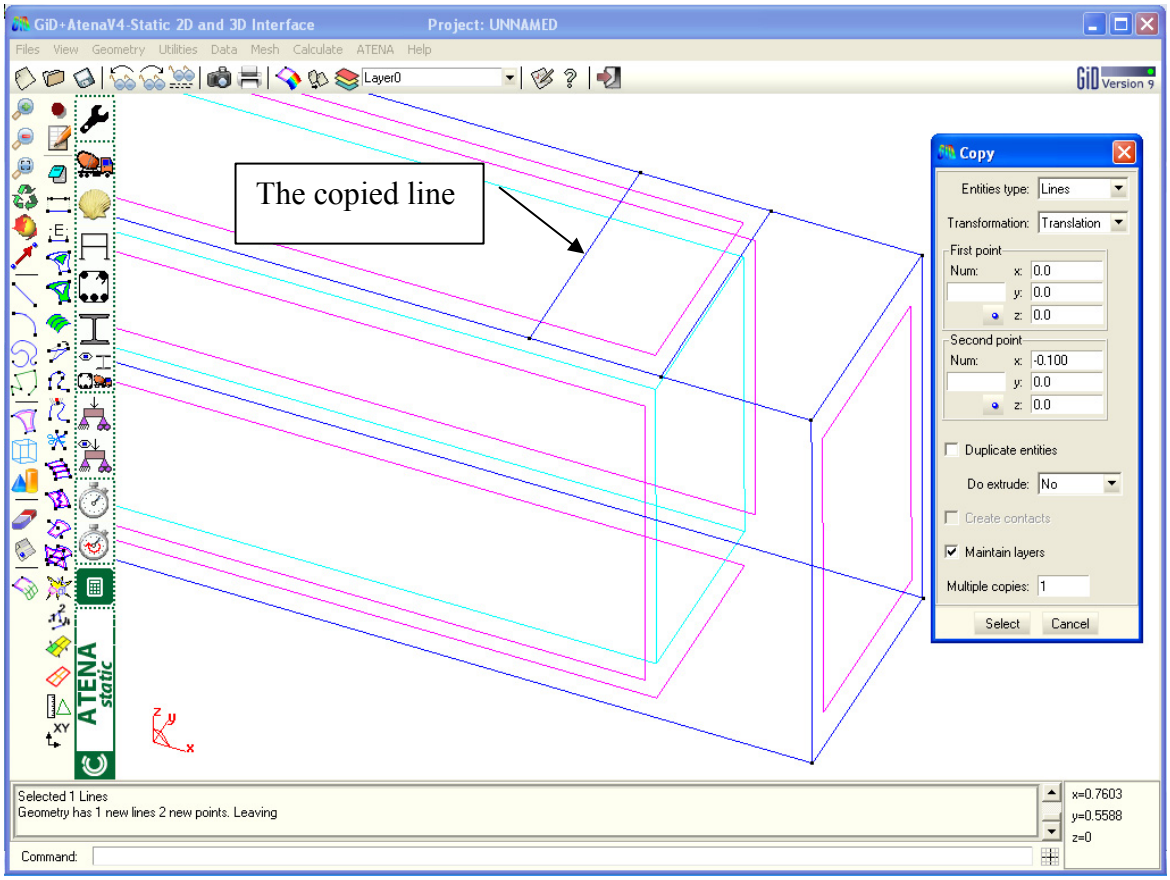


Figure 22: The repeated copy operation to create the second line

Next step is to connect these newly copied lines into a rectangle. This can be done by creation of new lines. Therefore it should be used command from the main menu **`Geometry | Create | Straight line`** or by clicking the icon . Also the **`Join`** function should be used (Ctrl + a; see chapter 3.2.1). The connection of lines is depicted in the Figure 23 and the Figure 24.

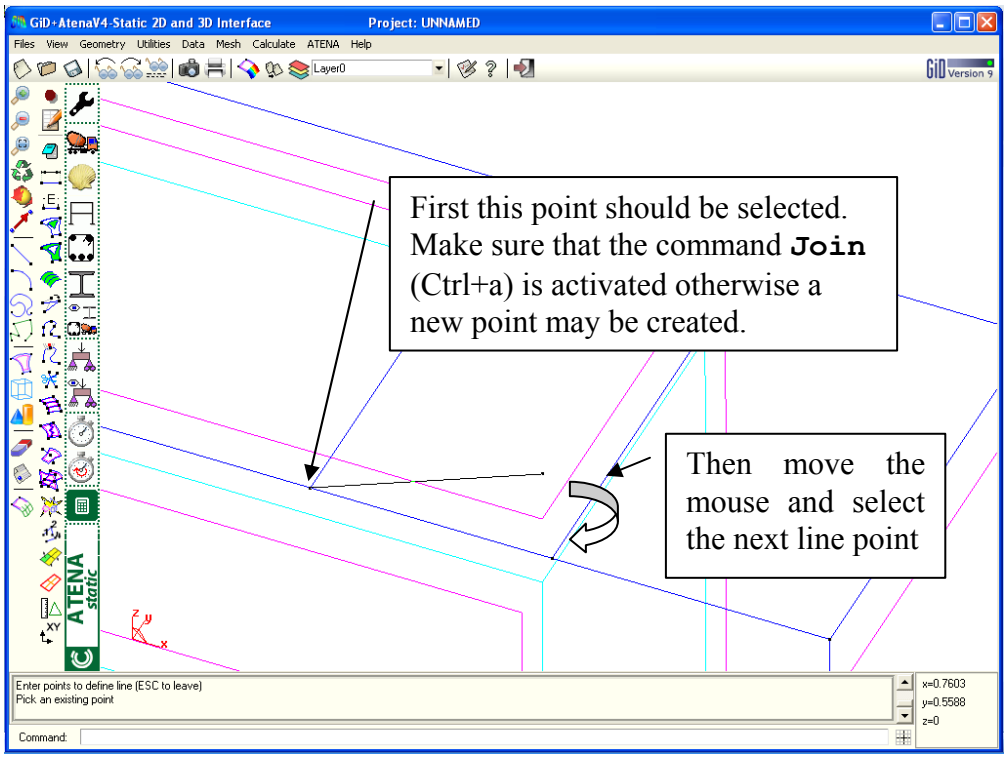

GiD+AtenaV4-Static 2D and 3D Interface
Project: UNNAMED
First this point should be selected. Make sure that the command Join (Ctrl+a) is activated otherwise a new point may be created.
Then move the mouse and select the next line point
Enter points to define line (ESC to leave)
Pick an existing point
Command:

Figure 23: The two lines need to be connected to form a rectangle. The creation of the bottom line

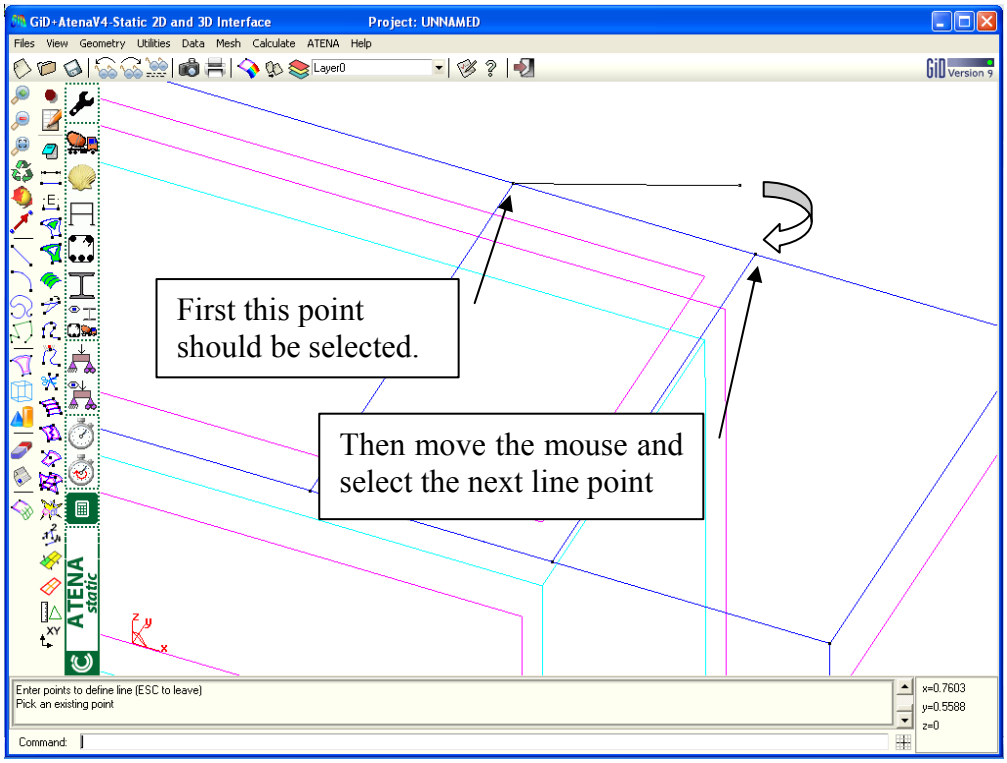

GiD+AtenaV4-Static 2D and 3D Interface
Project: UNNAMED
First this point should be selected.
Then move the mouse and select the next line point
Enter points to define line (ESC to leave)
Pick an existing point
Command:

Figure 24: The creation of the top line to finalize the rectangle for the bottom surface of the top plate.

After connecting lines into a rectangle, the surface should be created. For that it is useful to use an automatic surface definition with the command **`Geometry | Create | NURBS surface |Automatic`**. When this automatic method is used, the program asks for the number of bounding lines (see Figure 25). After definition of this number, the program automatically creates all possible surfaces with the given number of bounding lines.

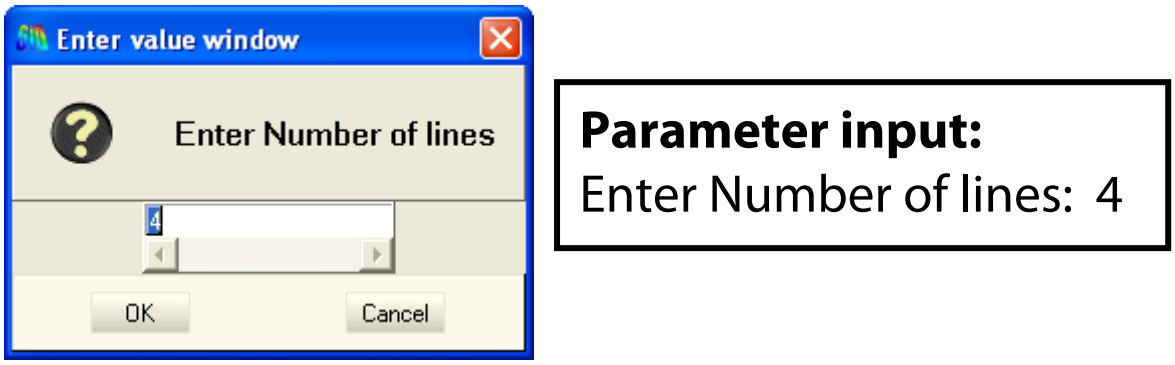


Figure 25: The definition of number of bounding lines

After clicking on the **`OK`** button, the required surface is created (see Figure 26). Then the button **`Cancel`** should be selected to leave this function.

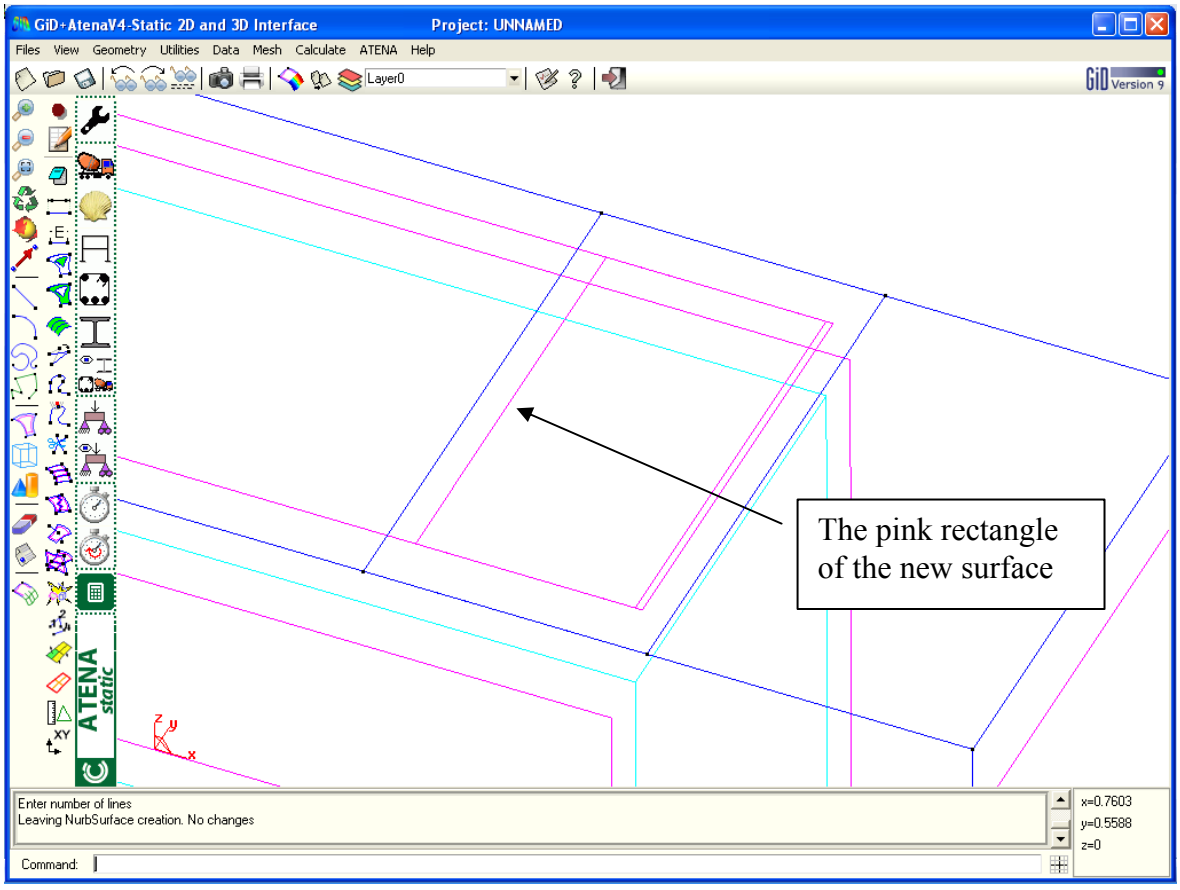
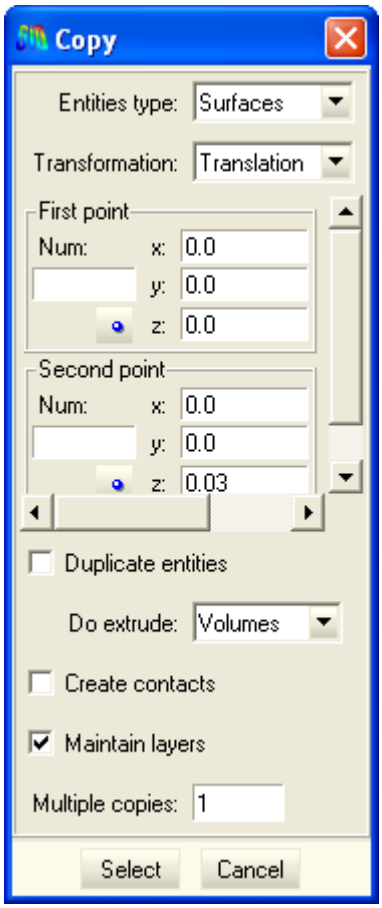


Figure 26: The surface created by automatic surface creation.

The geometry definition of the top plate will be finished by extrusion of the surface. The extrusion is done by the **Copy** command, which appears after selecting item from the main menu **Utilities | Copy**. The height of the steel plate is **0.030 m**. The definition of the extrusion is depicted in the Figure 27. After the definition of all parameters the **Select** button should be pressed. Then the surface required for the extrusion can be selected in the graphical area (see Figure 28). After the selection of surface it is necessary to press **Finish** button to complete the extrusion (see Figure 29).



Parameter input:
Entities type: Surfaces
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.0
z: 0.03
Do extrude: Volumes

Figure 27: The definition of the steel plate extrusion

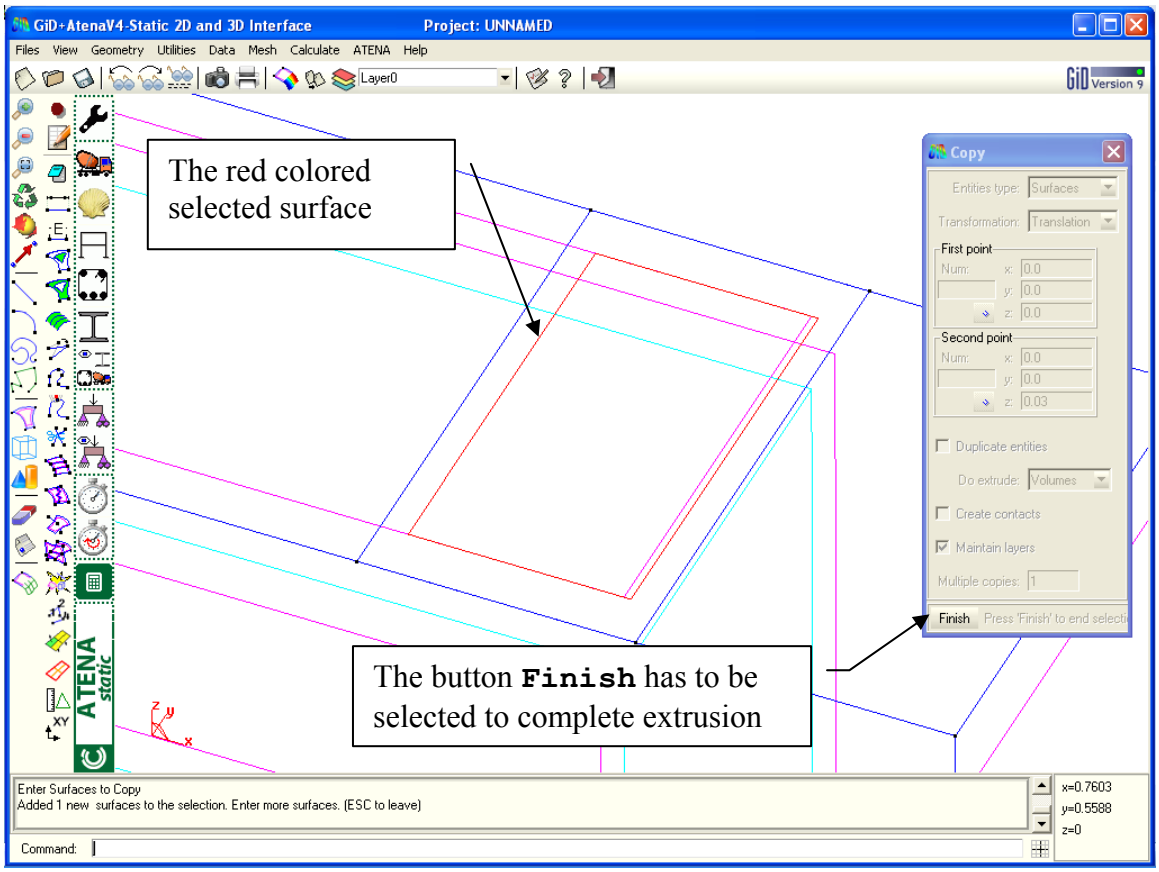


Figure 28: The selection of the surface which should be extruded to obtain steel plate geometry

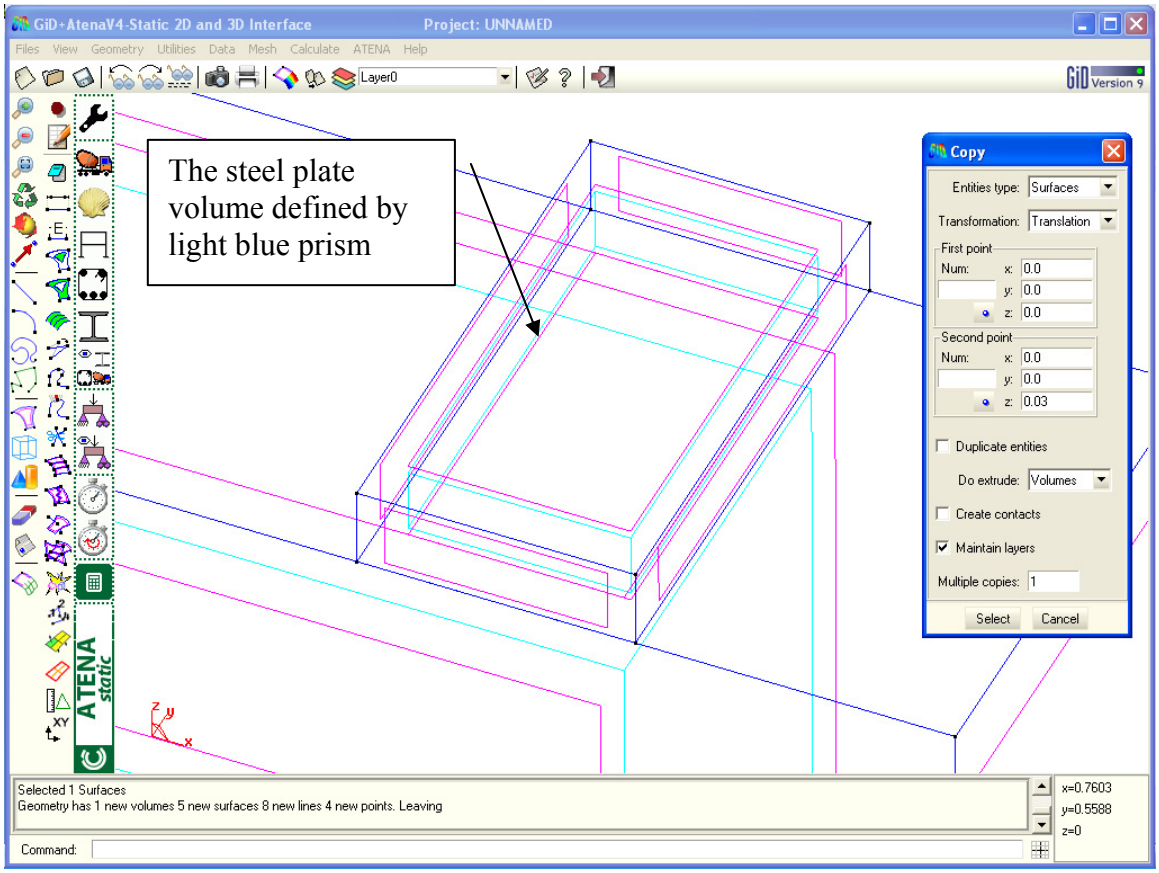


Figure 29: The volume of the top steel plate

3.2.2.2 Bottom plate

The bottom steel plate will be created by copying of the top plate.

The copy starts by command **`Utilities | Copy`** in the Main menu. The definition of the extrusion is depicted in the Figure 30. After the definition of all parameters the **`Select`** button should be pressed. Then the volume required for the translation can be selected in the graphical area (see Figure 31). After the selection of volume it is necessary to press **`Finish`** button to complete the translation (see Figure 32).

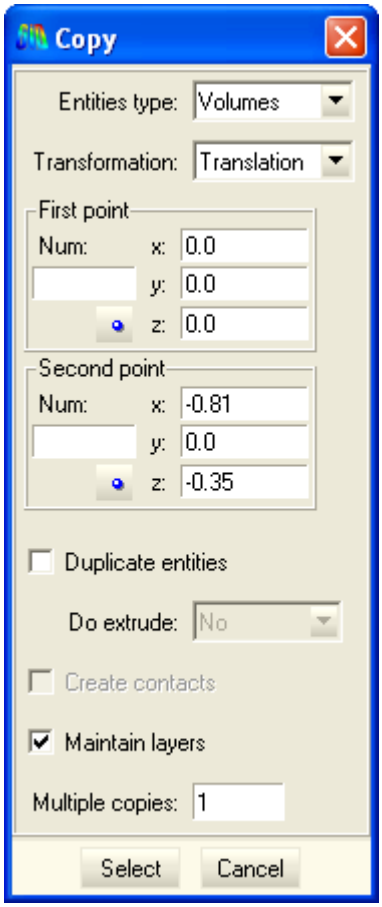


Figure 30: The parameter definition

Parameter input:
Entities type: Volumes
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: -0.81
y: 0.0
z: -0.35
Do extrude: No

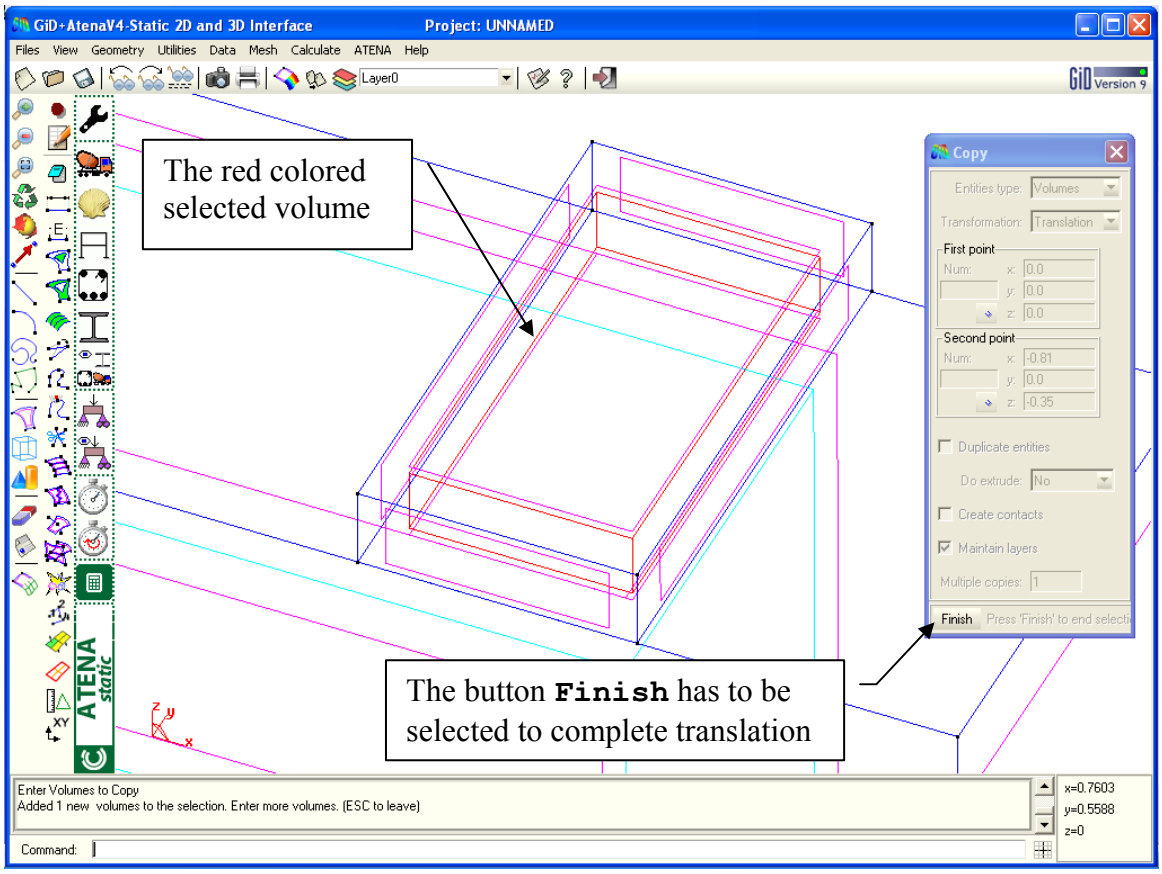


Figure 31: The selection of the volume which should be copied

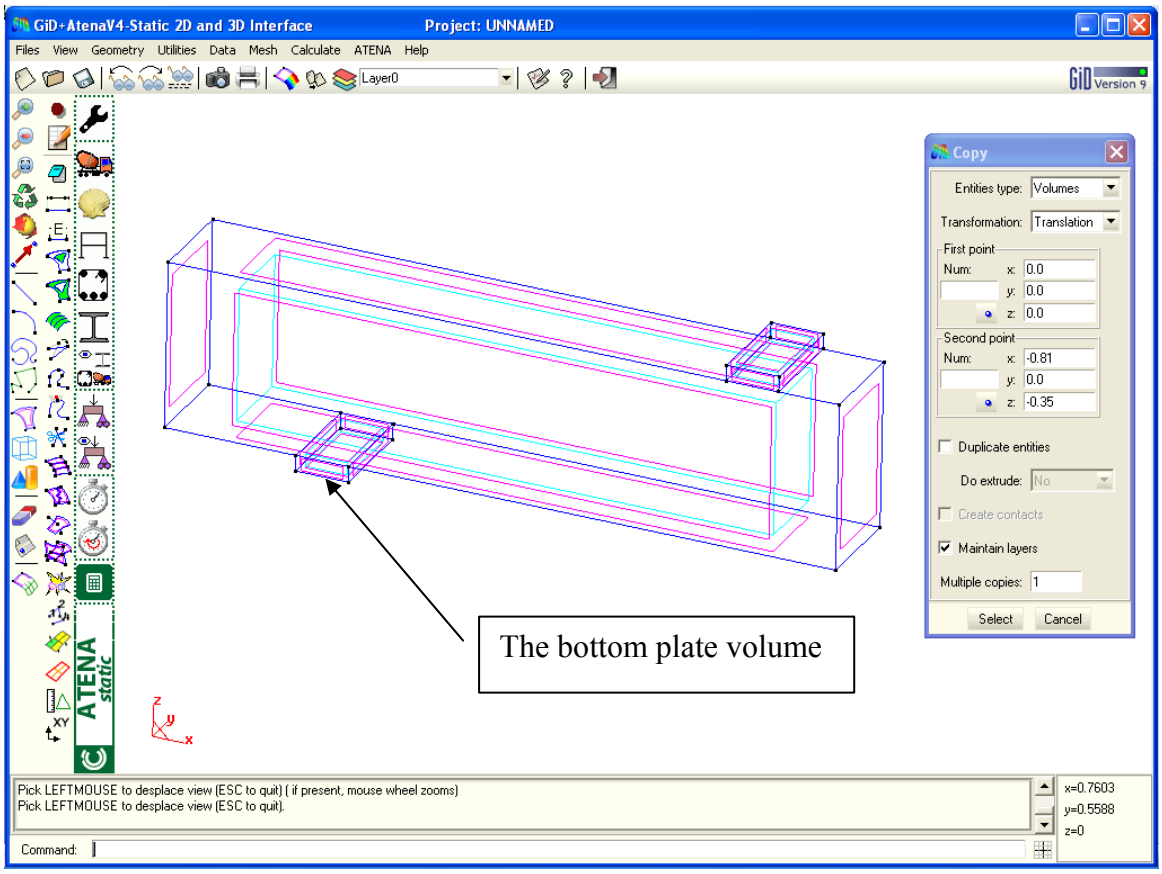


Figure 32: The bottom and top plates

3.2.3 Reinforcement bars

The geometry of reinforcement bars will be defined only by two lines. The first bar will be created and then the second bar will be copied.

The creation of the first bar starts by clicking the icon or with the command from the main menu **`Geometry | Create | Straight line`**. The command line in the bottom of the main window should be used for the coordinates' definition. The coordinates of the reinforcement are (**0.05,0.05,0.05**) and (**1.275,0.05,0.05**). See Figure 33.

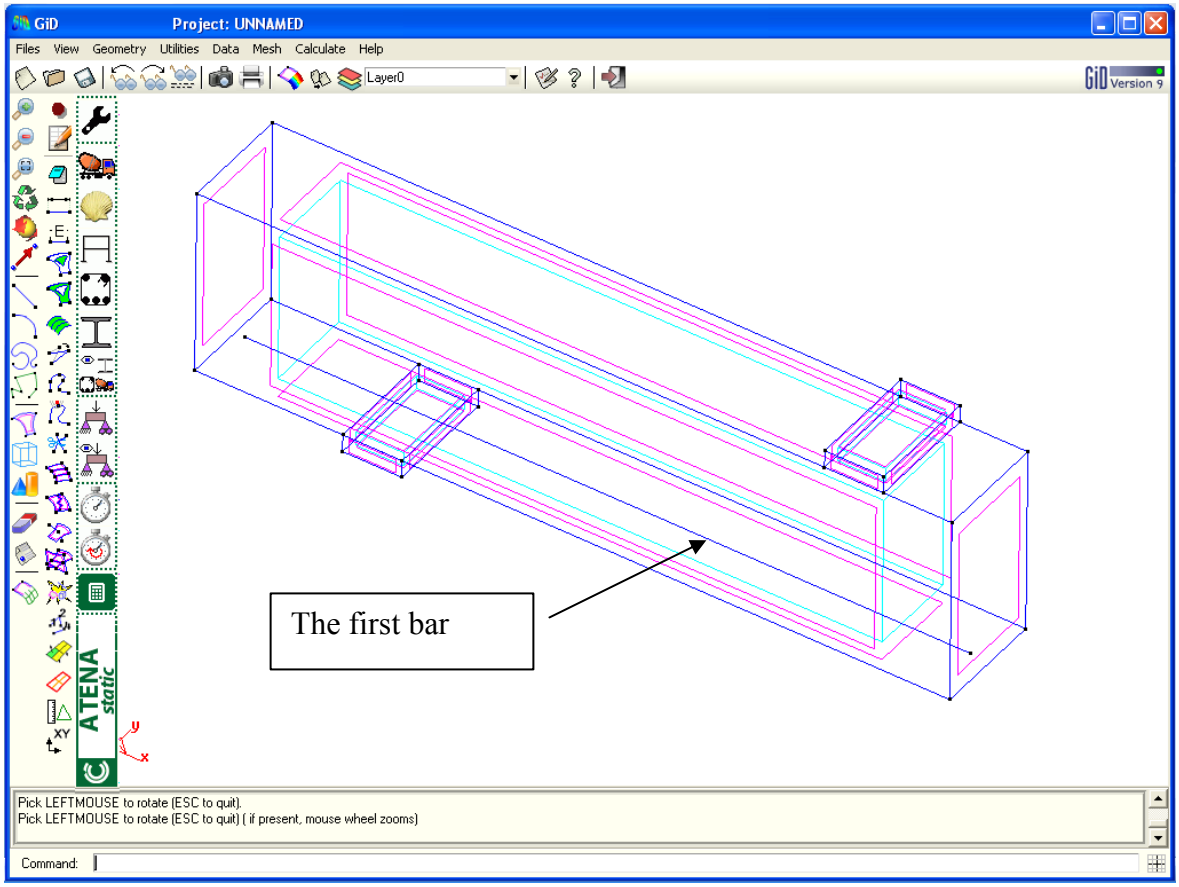
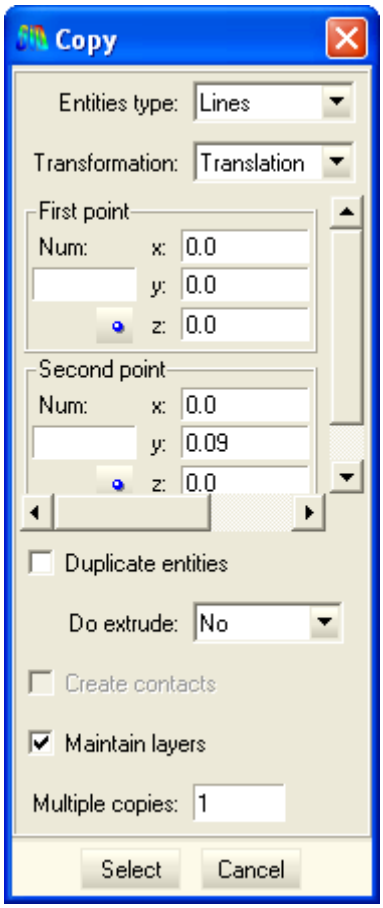

GiD
Project: UNNAMED
Files View Geometry Utilities Data Mesh Calculate Help
Layer0
GiD Version 9
The first bar
ATENA static
Pick LEFTMOUSE to rotate (ESC to quit).
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Command:

Figure 33: The first reinforcement bar

Parameter input:
Coordinates of the line:
1: 0.05,0.05,0.05
2: 1.275,0.05,0.05

Second reinforcement bar will be created by copying of the first bar. The copy starts by command **`Utilities | Copy`** in the main menu. The definition of the translation is depicted in the Figure 34. After the definition of all parameters the **`Select`** button should be pressed. Then the line required for the translation can be selected in the graphical area (see Figure 35). After the selection of line it is necessary to press **`Finish`** button to complete the translation (see Figure 36).



Parameter input:
Entities type: Lines
Transformation: Translation
First point: x: 0.0
y: 0.0
z: 0.0
Second point: x: 0.0
y: 0.09
z: 0.0
Do extrude: No

Figure 34: The parameter definition for the copying of the first bar

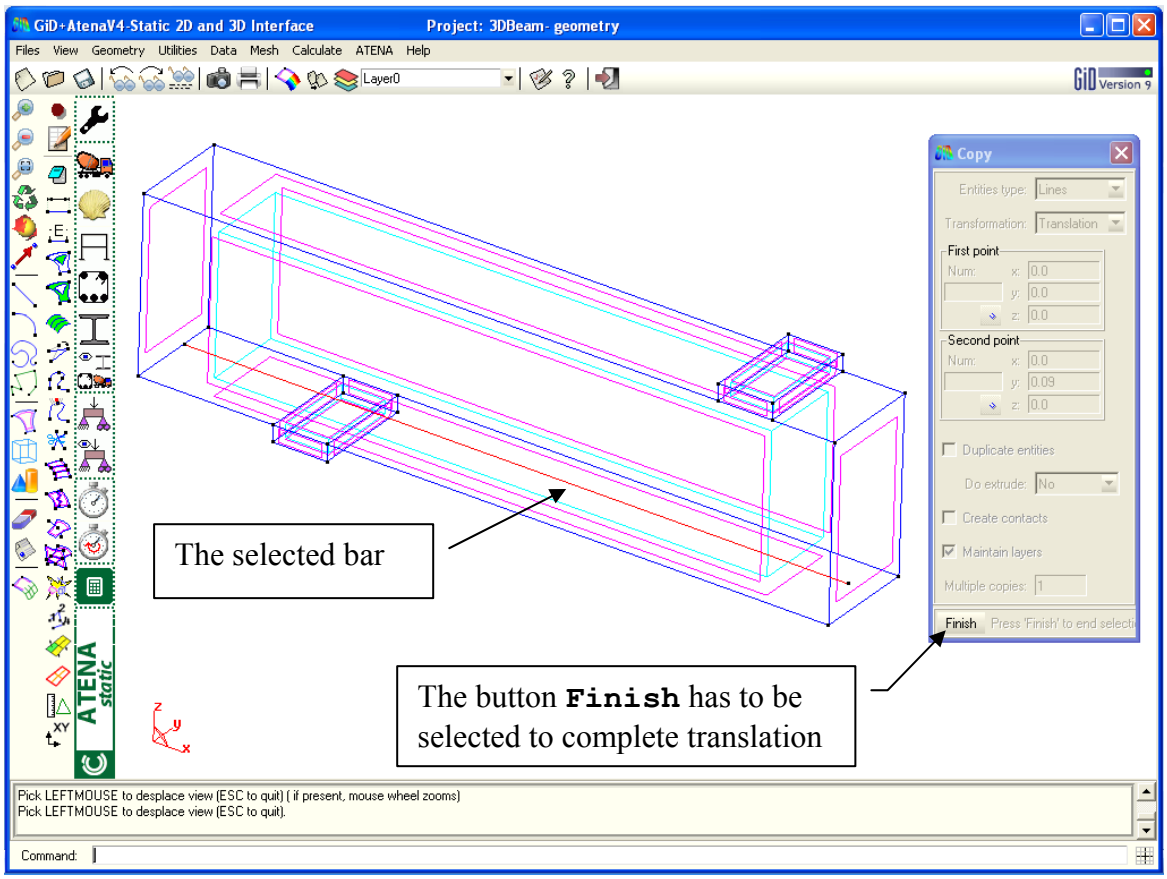


Figure 35: The selection of the first reinforcement bar which should be copied

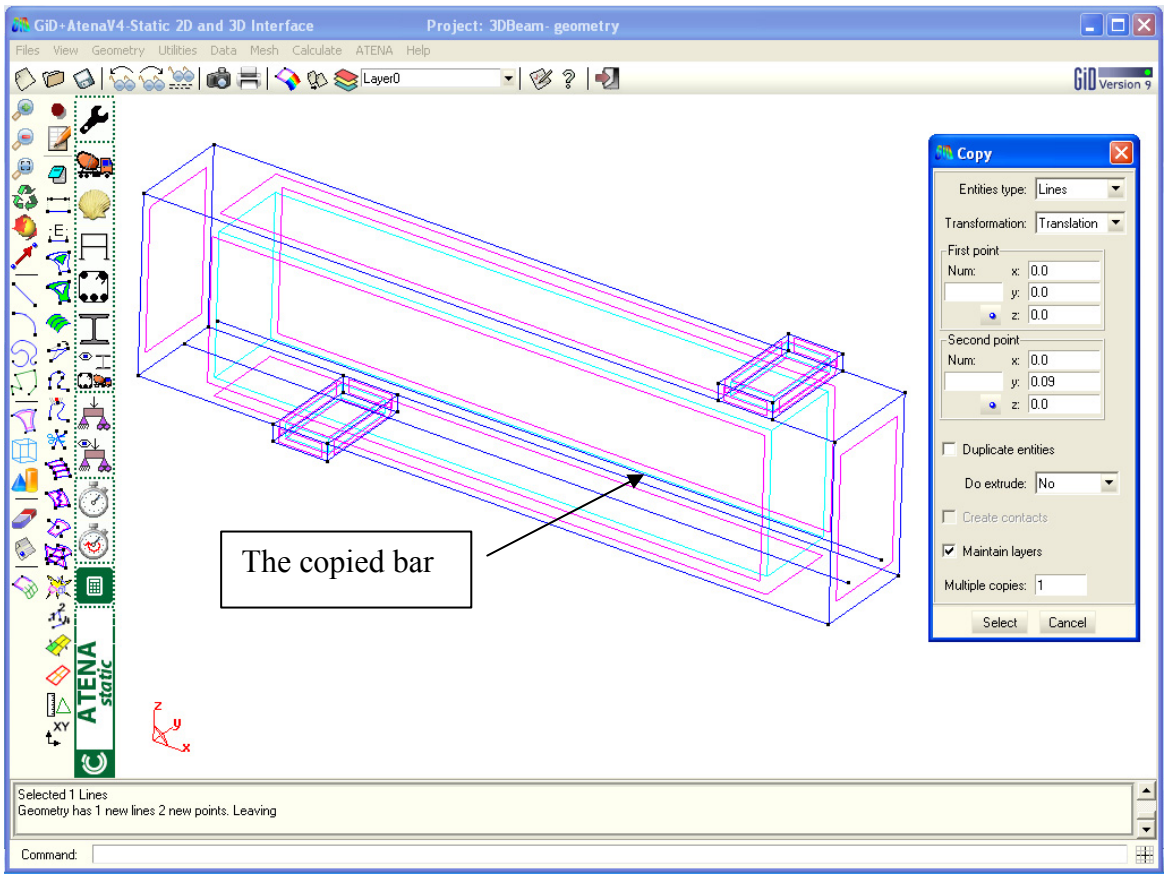


Figure 36: The first and second reinforcement bar

3.2.4 Layers

Layers are useful feature of **GiD**. The individual components of the created geometry can be separated into different layers. In each layer and its components can be shown and the user can easily work only with components of this layer.

In this chapter three different layers will be created – concrete beam layer, steel plates layer and reinforcement layer.

3.2.4.1 Beam layer

It is good to start with the definition of concrete beam layer. This is done by the command **`Layers`**, which appears after selecting **`Utilities | Layers`** in the main menu. The beam layer will be created by writing **beam** into a window depict on the Figure 37. The new layer will be created after the pressing of the button **`New`**. Then the beam layer will appear in the list of layers.

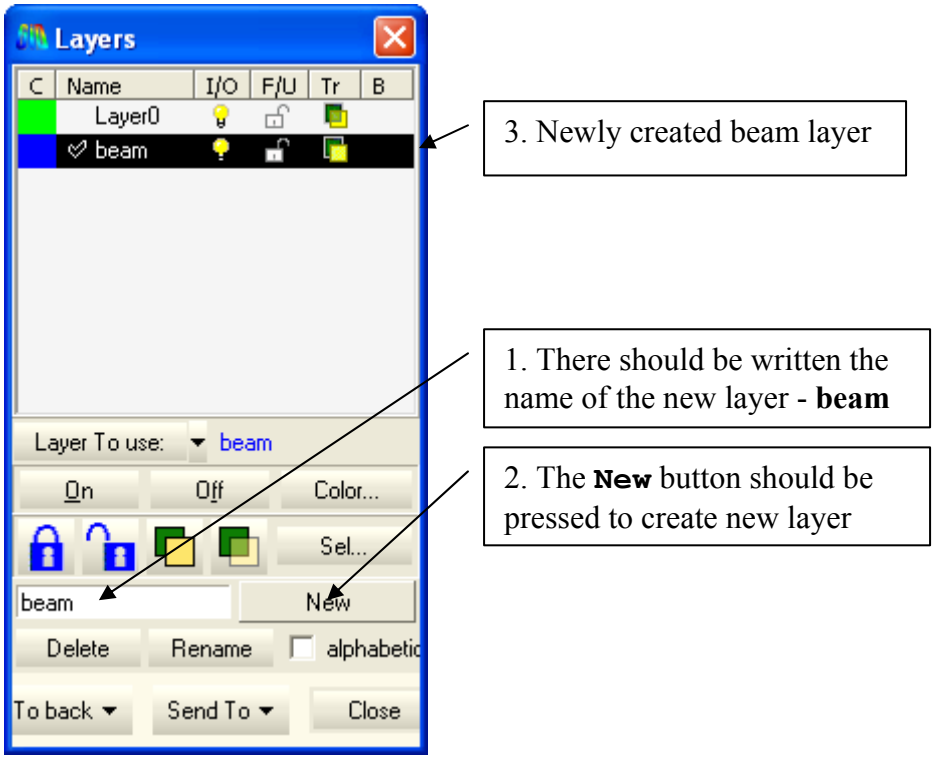


Figure 37: The Layers command

The newly created beam layer is immediately activated. The activation is sign by a ☑. The moving of the beam geometry to the beam layer can be started by pressing of the button **`Send to`**. Then the pull down menu will open (see Figure 38). The beam geometry contains three types of entities and all of them should be moved into the beam layer. Therefore the **`Also lower entities`** has to be activated and the command **`Volumes`** should be chosen. After selecting the **`Volumes`** in the pull down menu the geometry, which should be send to the beam layer, can be selected (see Figure 39). The pressing **`Finish`** button will complete this command.

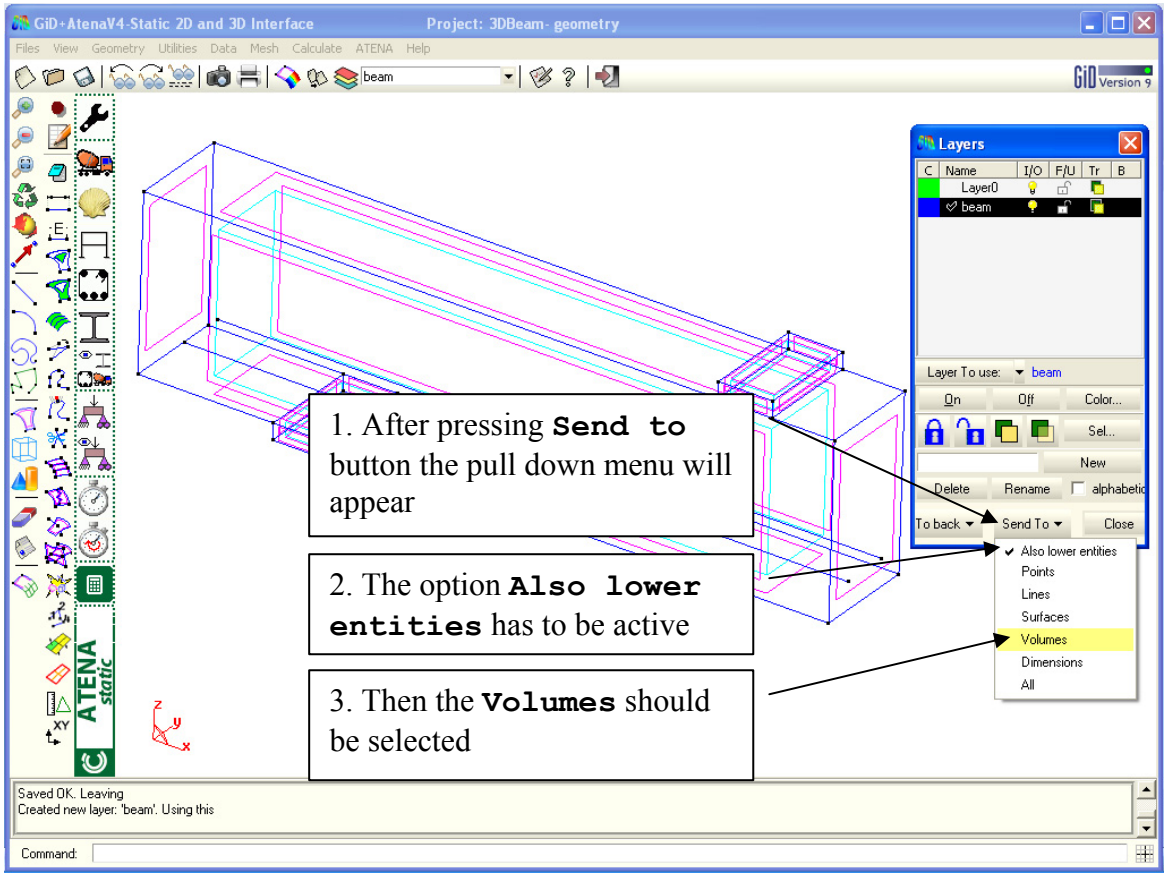


Figure 38: The definition of the `Send to` command for the beam layer

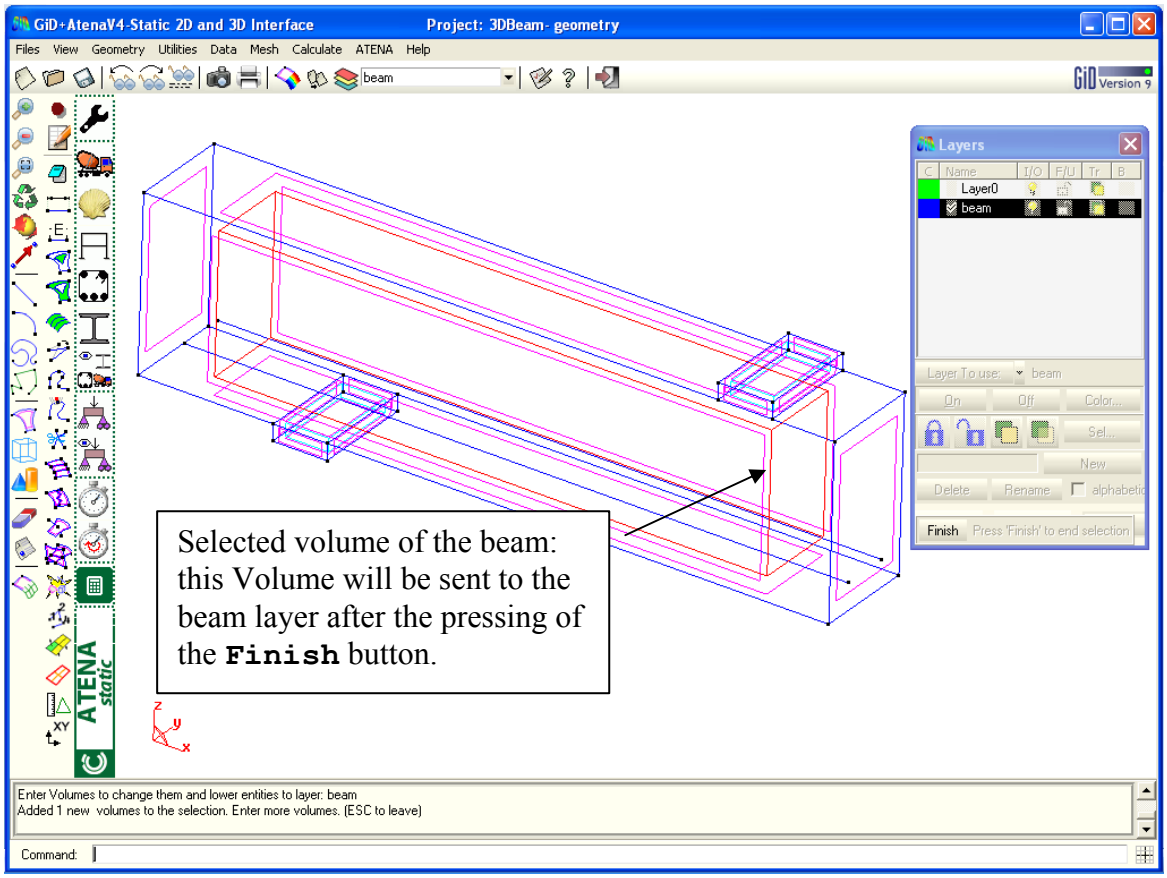

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- geometry
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
Layers
C Name I/O F/U Tr B
Layer0
beam
Layer To use: beam
On Off Color...
Sel...
New
Delete Rename alphabetic
Finish Press 'Finish' to end selection
Selected volume of the beam: this Volume will be sent to the beam layer after the pressing of the Finish button.
ATENA static
Enter Volumes to change them and lower entities to layer: beam
Added 1 new volumes to the selection. Enter more volumes. (ESC to leave)
Command:

Figure 39: The selection of the volume, which should be sent to the beam layer

By the clicking selecting button **On** or **Off** the content of the chosen layer can be seen or hidden. The yellow bulb next to the name of the beam represents the displaying of the content of the layer. Also direct clicking on the bulb of the layer can make this command. The Layer0 (the layer which was already there before creating the beam layer) should be selected and then the button **Off**. The yellow bulb will change to the grey colour . It means that all its content should not be displayed. The Layer0 still contains the geometry of steel plates and reinforcement. Therefore these geometries should disappear in the graphical area after deactivating of the Layer0 (see Figure 40). It should be possible to see only the beam and it assures that the beam geometry was successfully sent to the beam layer.

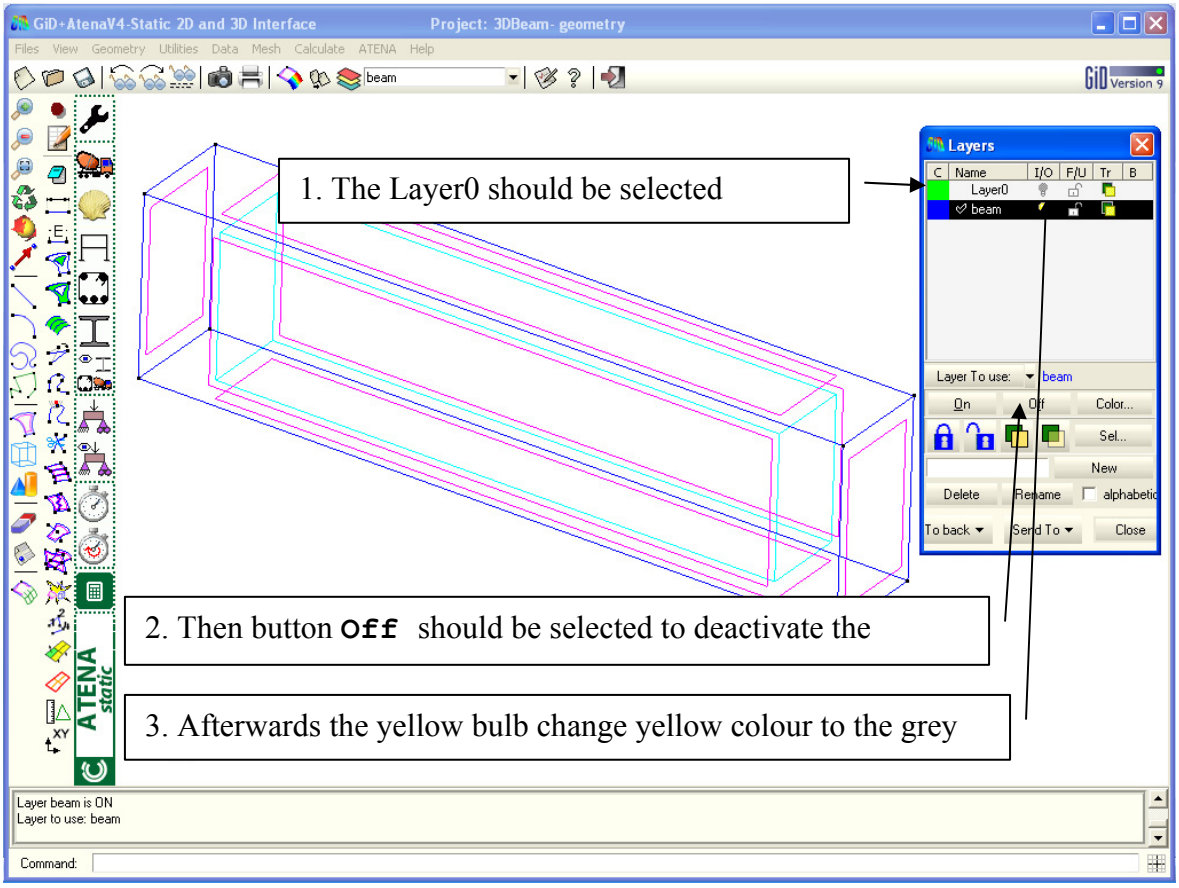


Figure 40: The steel plates and reinforcement geometry will disappear after deactivating of the Layer0

3.2.4.2 Bars layer

The next step is to create a bars layer. This layer will be created with the same procedure like in previous beam layer.

First the beam layer should be hidden and Layer0 should be displayed. It is done by selecting the beam layer and pressing button **Off** and Layer0 is displayed by selecting this layer in the list and then by pressing button **On**. Afterwards the beam geometry will disappear and the reinforcement and steel plates will appear in the graphical area (see Figure 41).

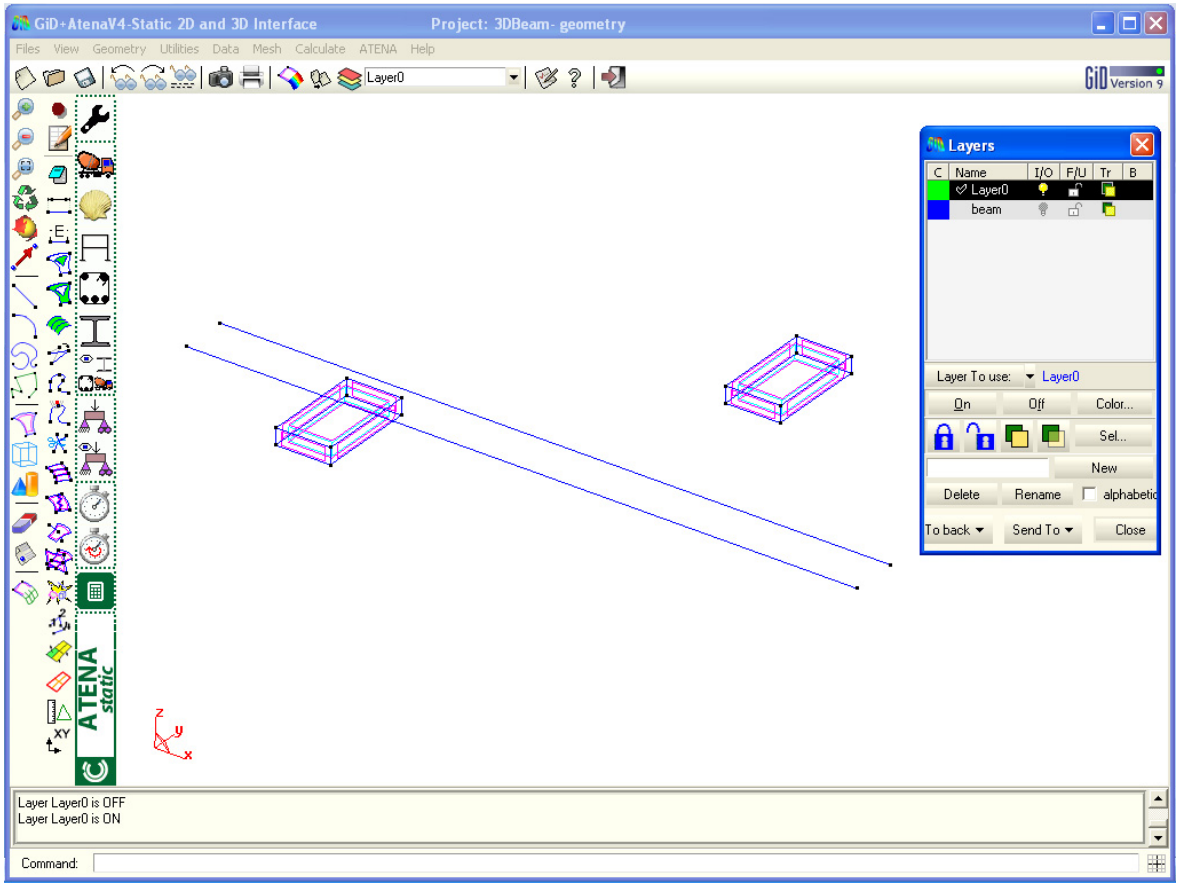


Figure 41: The Layer0 is activated and reinforcement and steel plates will appear in the graphical area

The reinforcement layer is created by writing the name **bars** into the window and pressing button **New**. Then the reinforcement layer will appear in the list of layers. The newly created bars layer is automatically activated. The activation is sign by a ☑.

The moving of the reinforcement geometry into the bars layer can be started by pressing of the button **Send to**. Then the pull down menu will open (see Figure 42). The reinforcement geometry contains two types of entities and all of them should be moved into the bars layer. Therefore the **Also lower entities** have to be activated and the command **Lines** should be chosen.

After selecting the **Lines** in the pull down menu the geometry, which should be send to the bars layer, can be selected (see Figure 43). The pressing **Finish** button will complete this command.

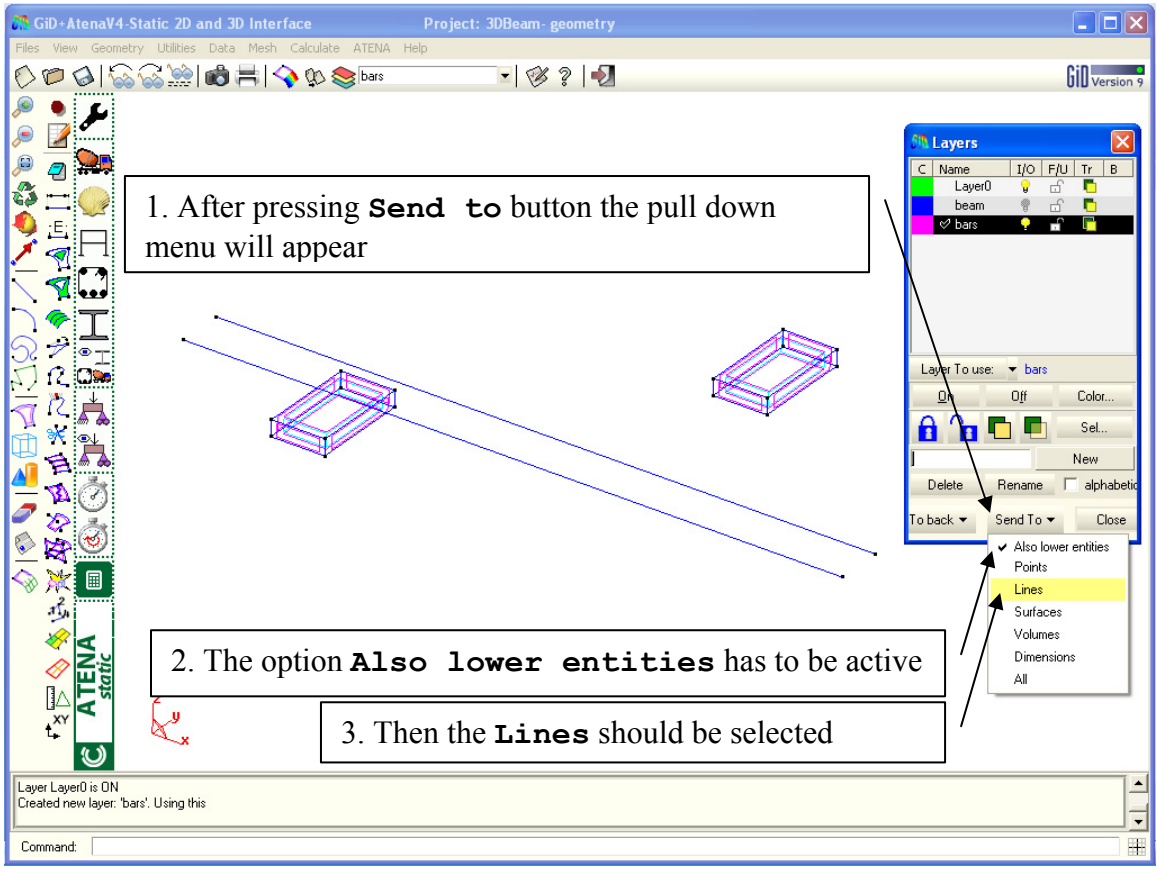


Figure 42: The definition of `Send to` command for the reinforcement layer

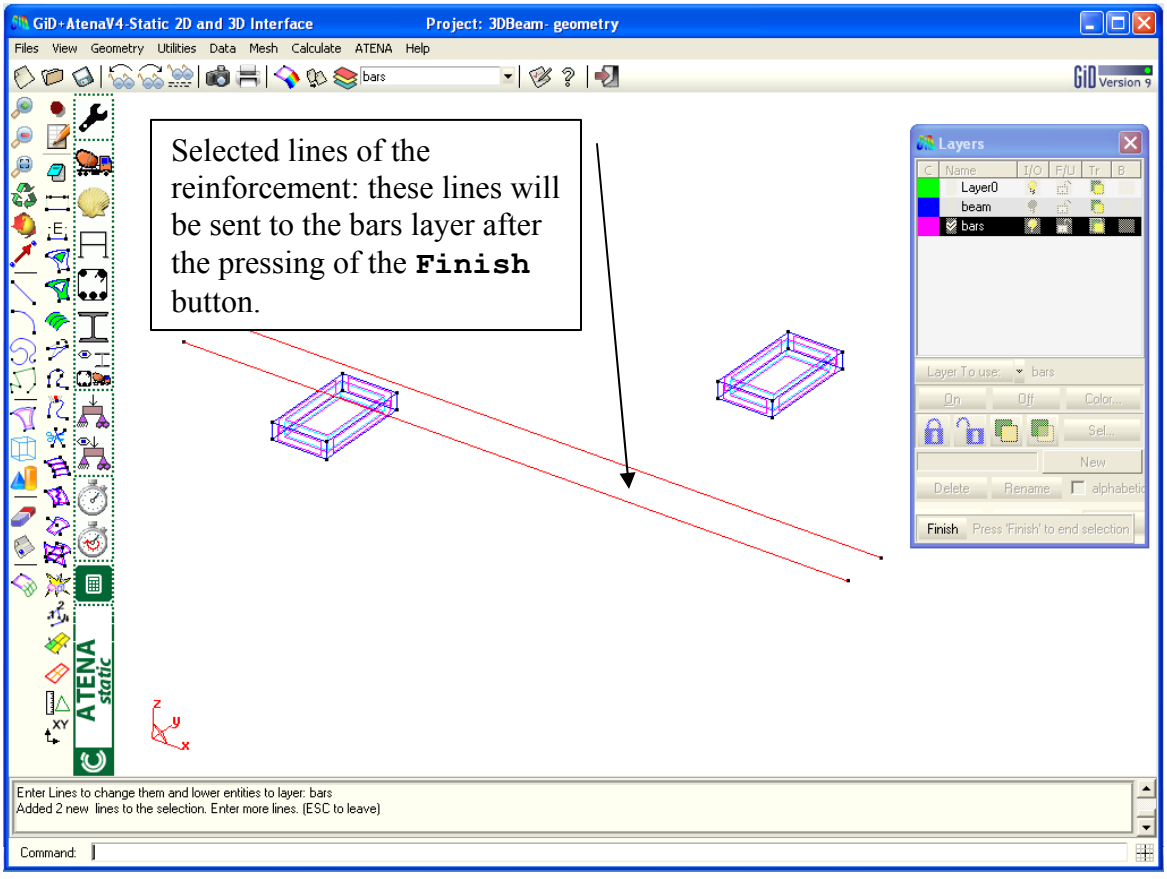

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- geometry
Selected lines of the reinforcement: these lines will be sent to the bars layer after the pressing of the Finish button.
Layers
Enter Lines to change them and lower entities to layer: bars
Added 2 new lines to the selection. Enter more lines. (ESC to leave)

Figure 43: The selection of the lines, which should be sent to the bars layer

3.2.4.3 Plates layer

After deactivating of the display of the bars layer, the lines should disappear. Deactivation is done by selecting the bars layer in the list of layers and then pressing the button **Off** (see Figure 44).

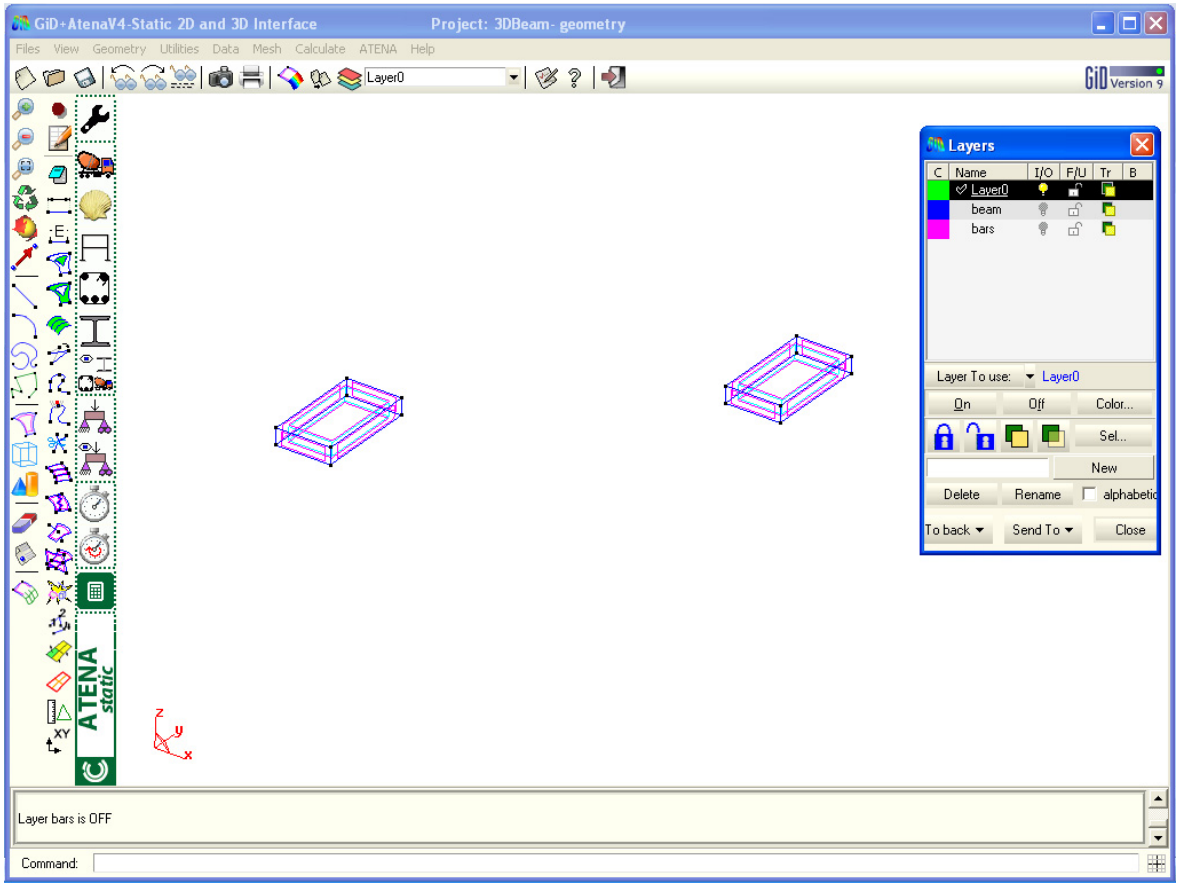


Figure 44: The reinforcement disappear after deactivating of the reinforcement layer

The last step is to create plates layer. Like in previous two layers it is done by writing the name **plates** into the window and pressing button **New**. Then the plates layer will appear in the list of layers. The newly created plates layer is automatically activated. The activation is sign by a ☑.

The moving of the steel plates geometry into the plates layer can be started by pressing of the button **Send to**. Then the pull down menu will open (see Figure 45). The reinforcement geometry contains two types of entities and all of them should be moved into the bars layer. Therefore the **Also lower entities** have to be activated and the command **Lines** should be chosen.

After selecting the **Lines** in the pull down menu the geometry, which should be send to the bars layer, can be selected (see Figure 46). The pressing **Finish** button will complete this command.

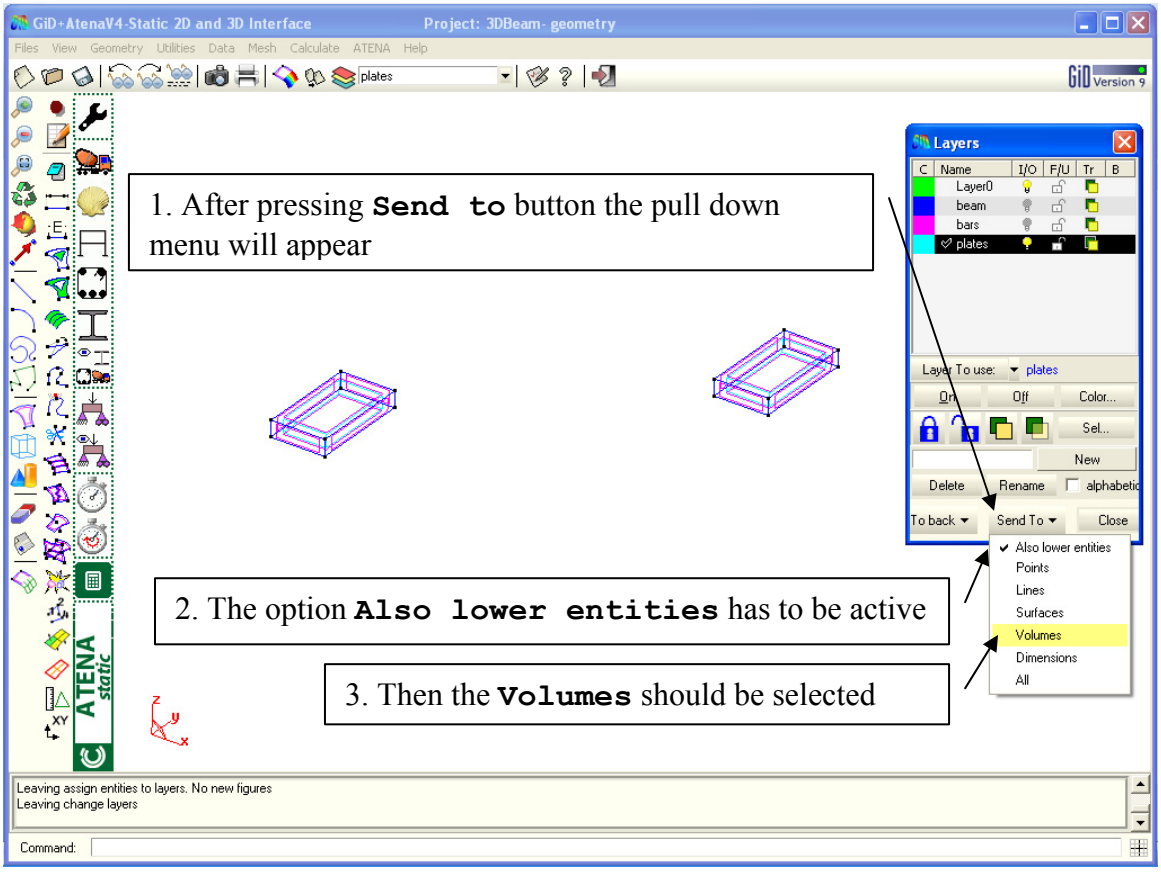


Figure 45: The definition of `Send to` command for the plates layer

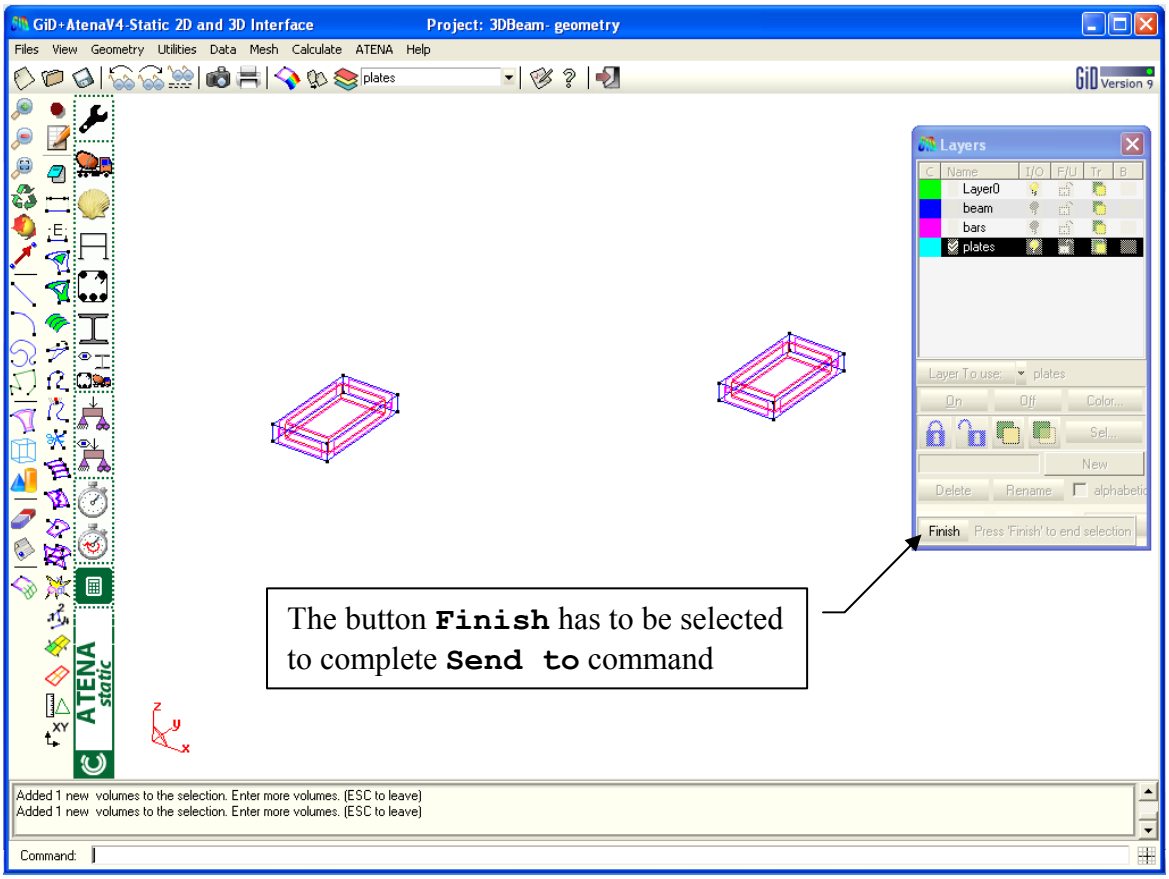

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- geometry
The button Finish has to be selected to complete Send to command
Layers
Finish
Added 1 new volumes to the selection. Enter more volumes. (ESC to leave)
Added 1 new volumes to the selection. Enter more volumes. (ESC to leave)
Command:

Figure 46: The selection of the volumes, which should be sent to the plates layer

After deactivating of the display of the plates layer, the volumes of the steel plates should disappear. Deactivation is done by selecting the plates layer in the list of layers and then pressing the button **Off** (see Figure 44).

The Layer0, which is now active, is empty. It does not contain any geometry and therefore this layer can be deleted. It is done by selecting this Layer and by pressing the button **Delete**. After that the Layer0 will be deleted (see Figure 47).

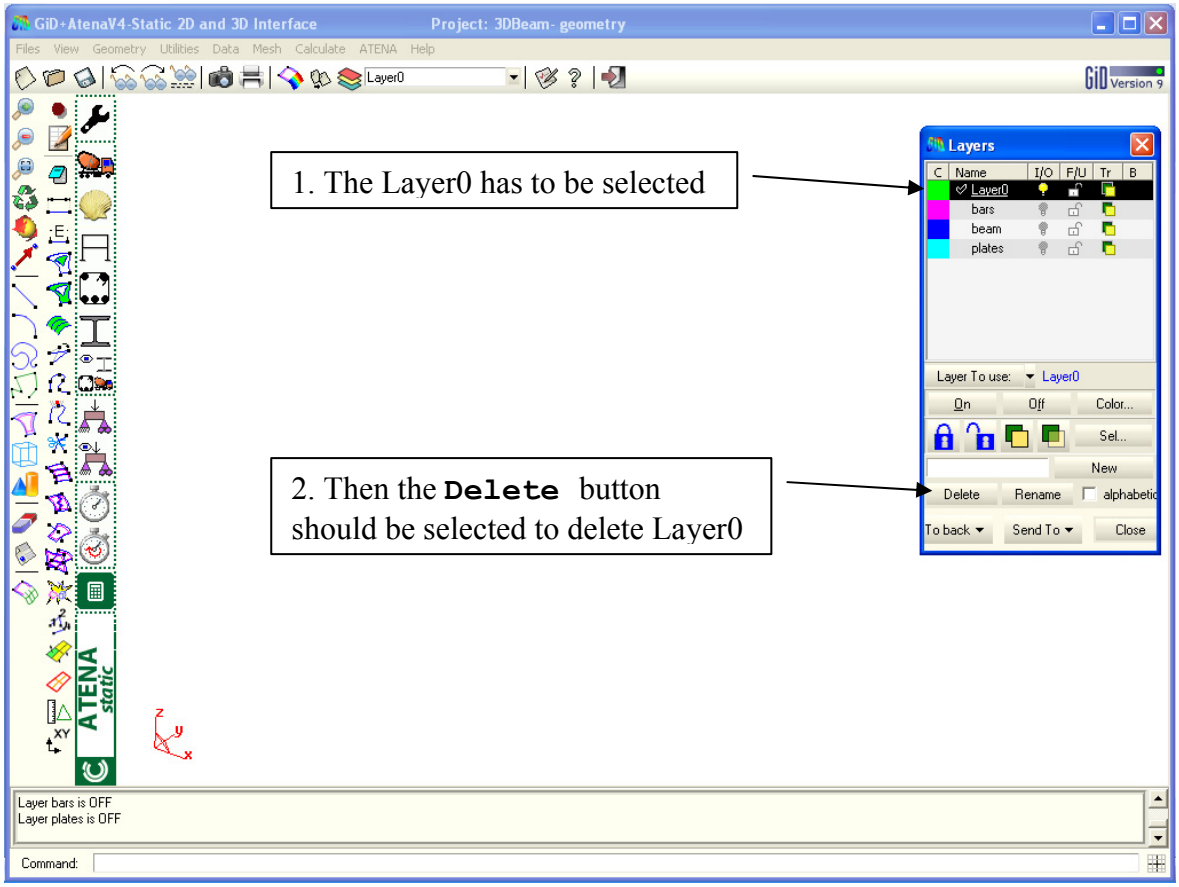


Figure 47: After deactivation of the plates layer the graphical area will stay empty. The Layer0 is active and it does not contains any geometry therefore it can be deleted.

It is recommended to display each layer separately to control, if they contain all required geometry. For the control see Figure 48, Figure 49and Figure 50.

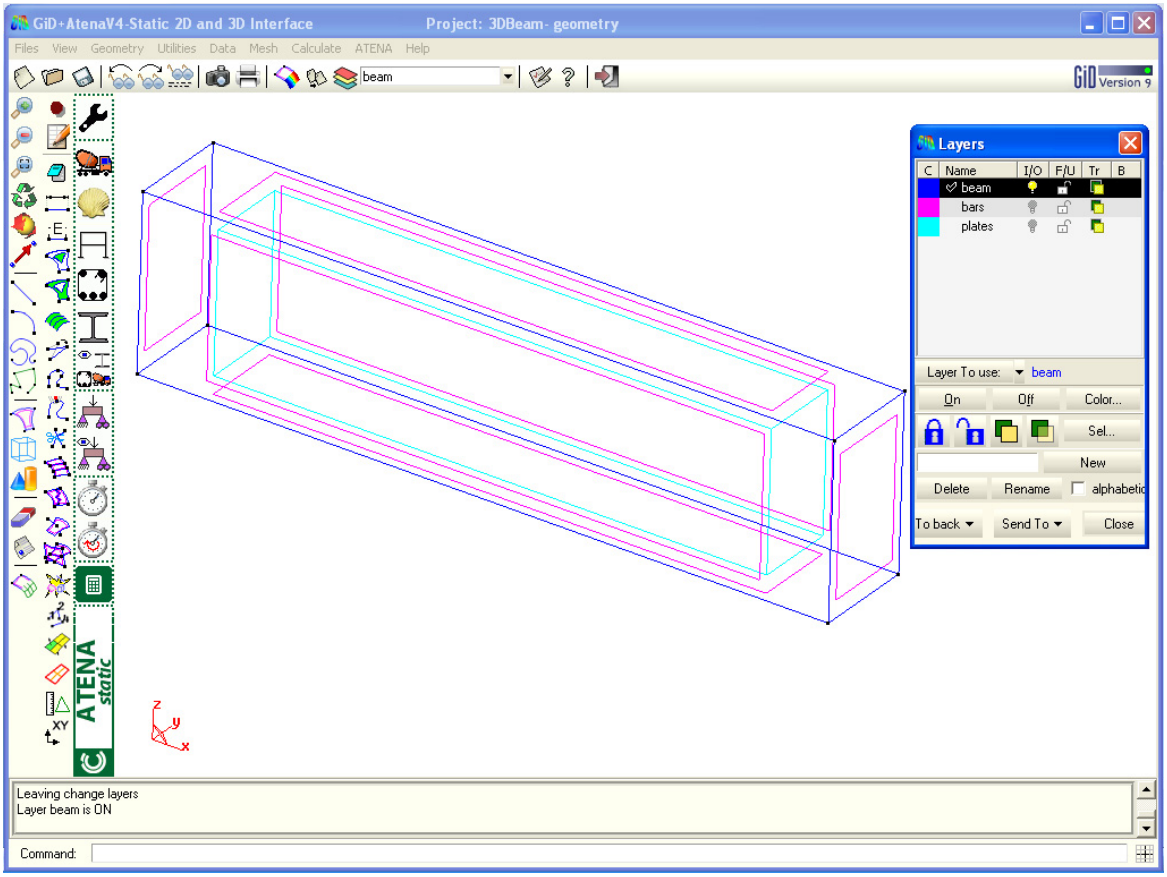


Figure 48: The displayed beam layer – contains beam volume

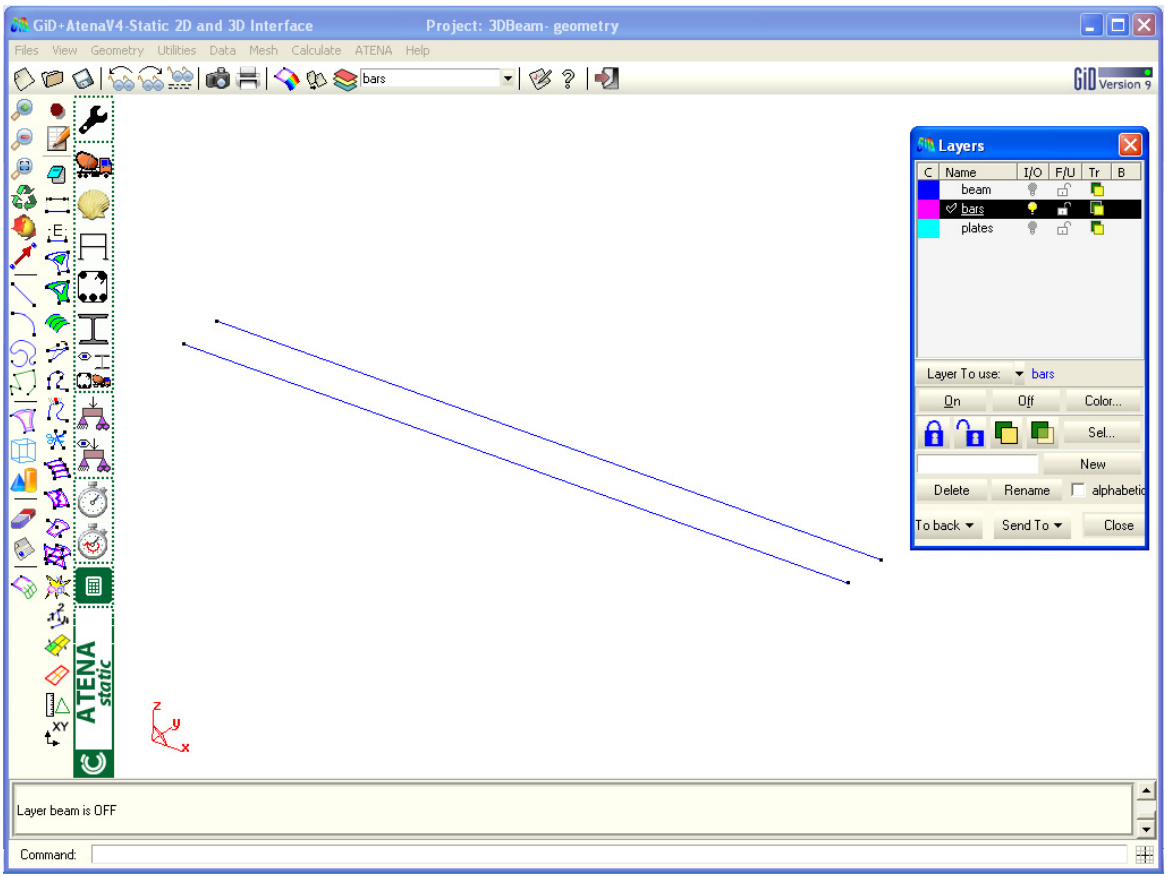


Figure 49: The displayed bars layer – contains reinforcement lines

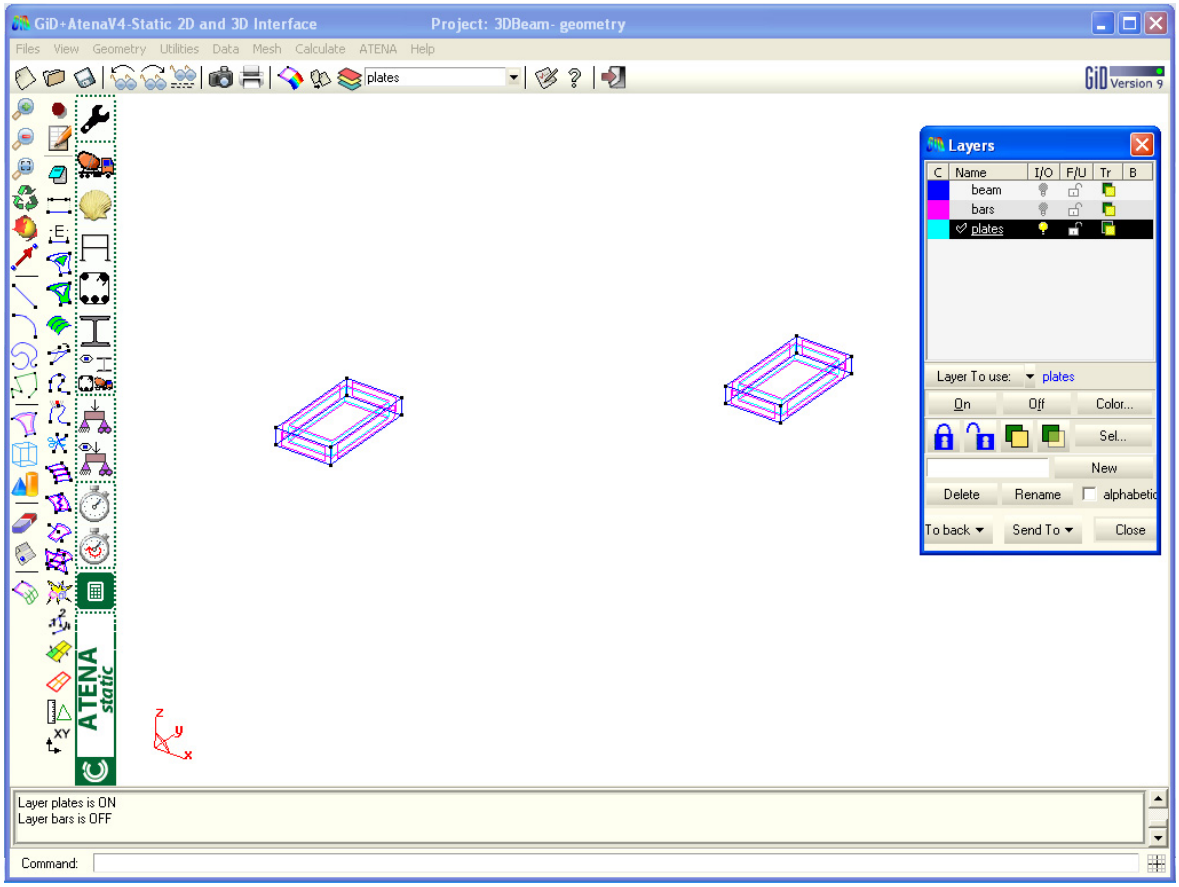


Figure 50: The displayed plates layer – contains plates volumes

3.3 Material parameters

This tutorial example contains three regions which are made from three different materials. These three regions are concrete beam, steel plates and reinforcement bars. In this chapter the characteristics of materials will be defined and then the material will be assigned to appropriate geometry.

3.3.1 Concrete beam

Before definition of the concrete beam material it is good to display only the beam layer.

The material definition of the beam starts by selecting of the icon or with the command **`Data | Materials | SOLID Concrete`** in main menu (see Figure 51).

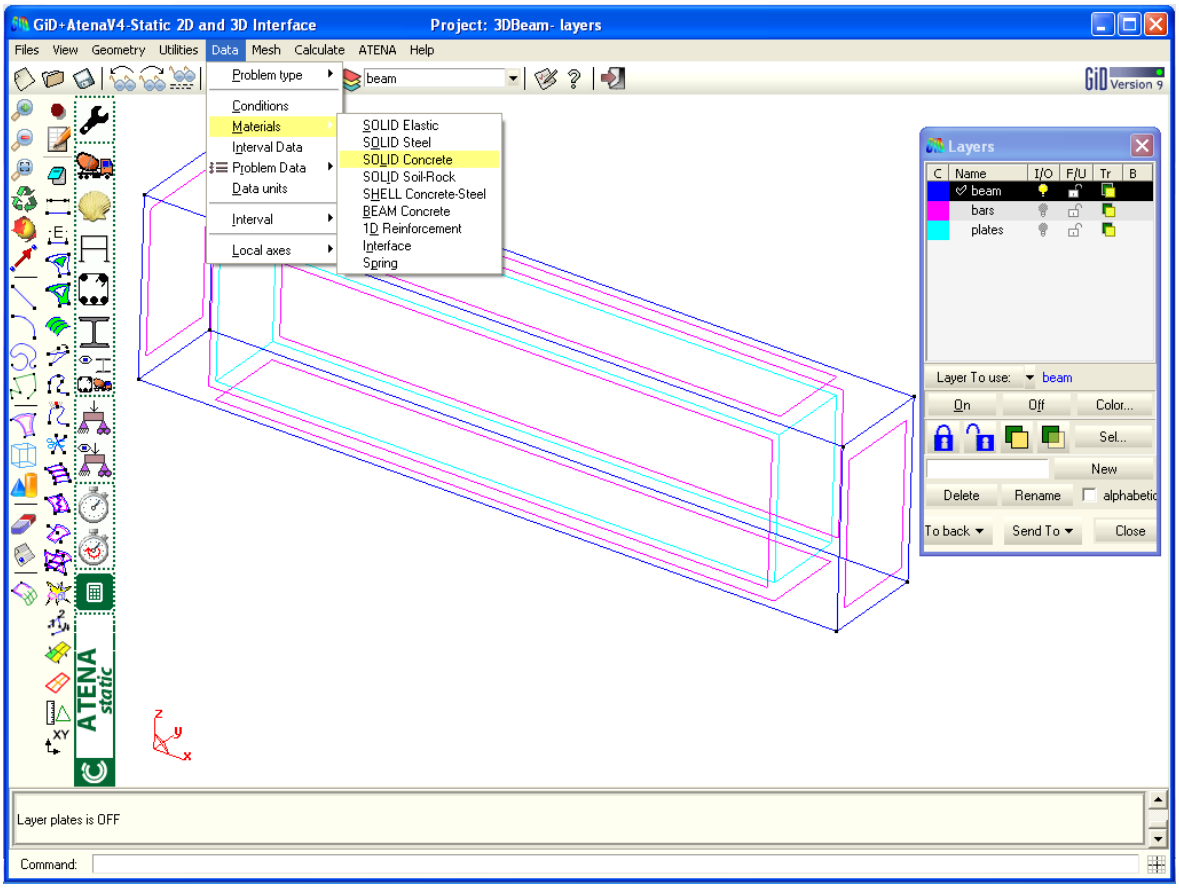


Figure 51: The selection of the command for the definition of the concrete material

After the selection of this command the window for the definition of the SOLID Concrete will appear (see Figure 52).

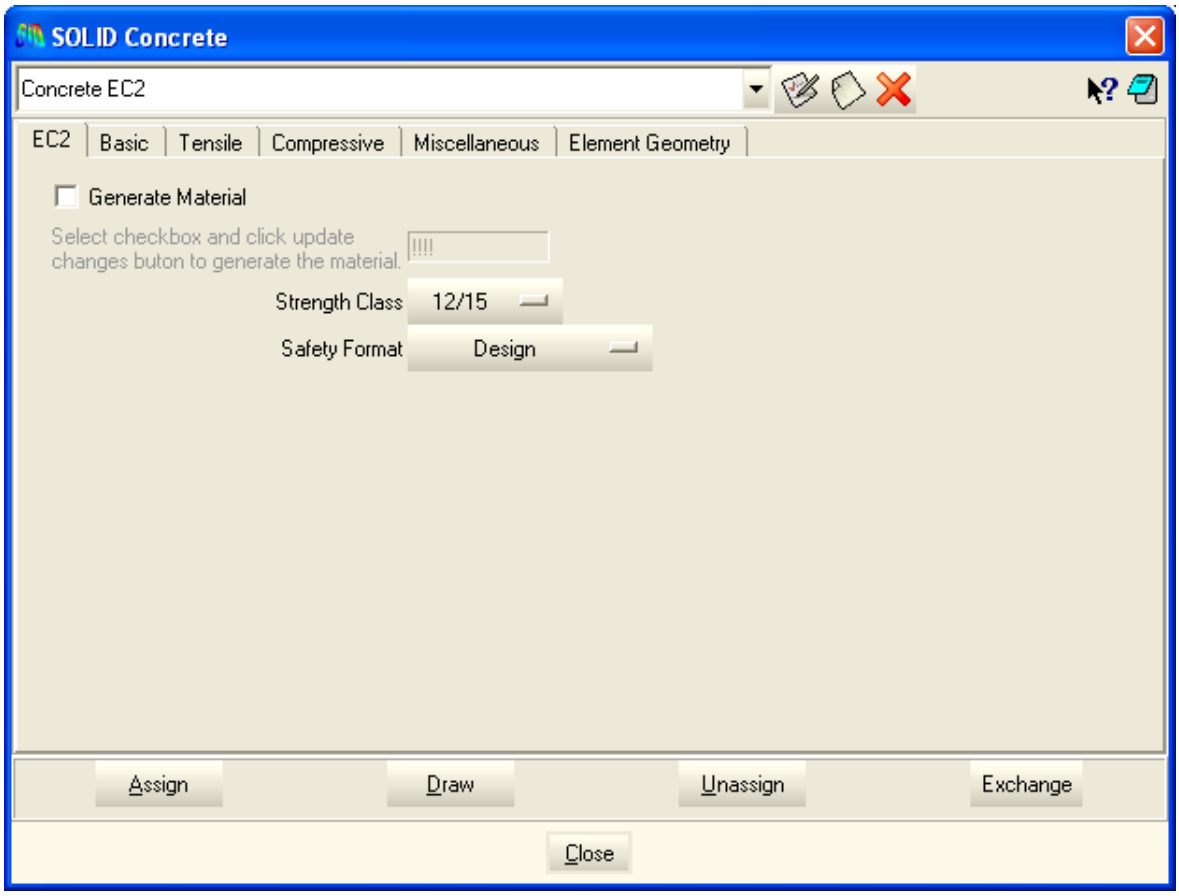


Figure 52: The window for the definition of the SOLID Concrete

First, it is important to copy material definition of the already existing material and save it under the new name. In this case the new name will be **Beam**. The predefined material Concrete EC2 should be chosen for the copying. After the selection of the predefined material the icon New SOLID Concrete should be selected.

The selection of this material and selection of the New SOLID Concrete icon are depicted in the Figure 53. After the selection of the New SOLID Concrete icon the new window for the definition of the new material name will appear (see Figure 54). There the **Beam** name should be written and then it is necessary to press **OK** button to complete this command.

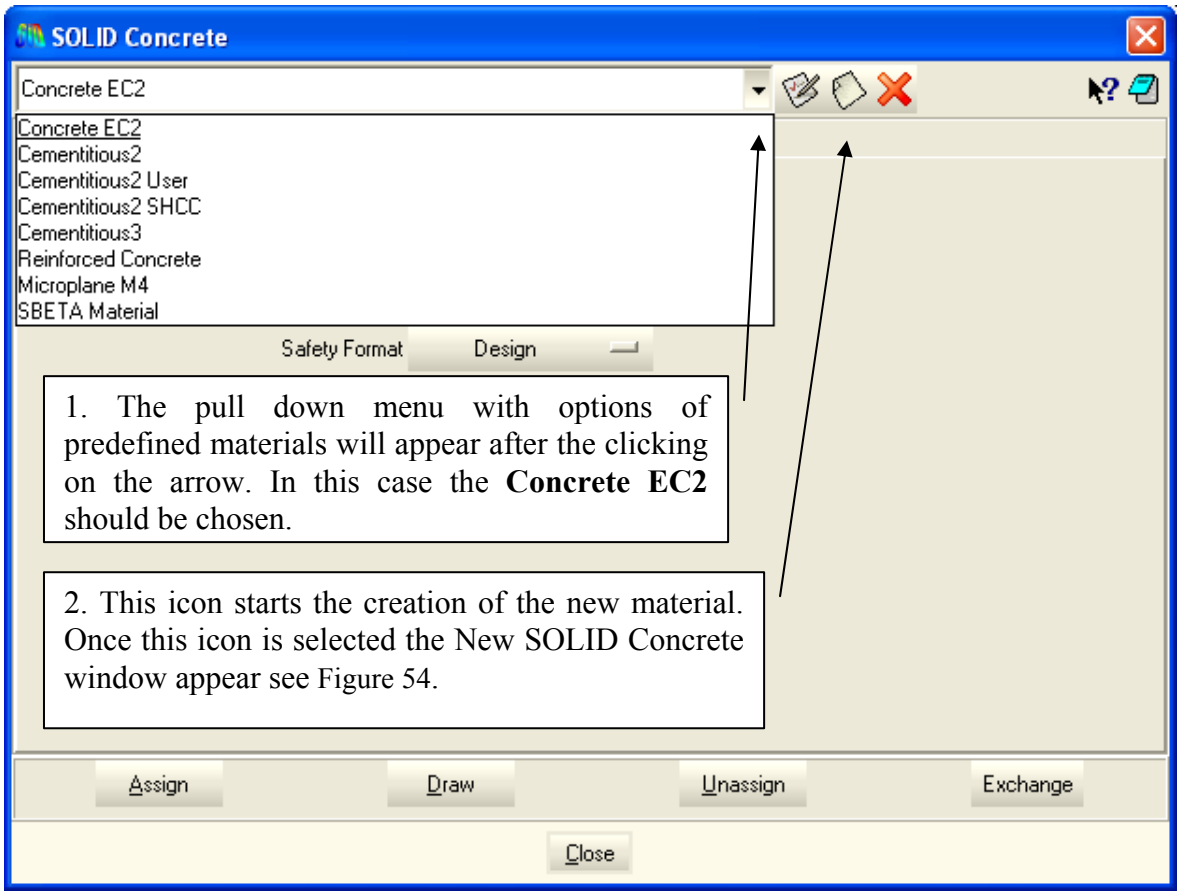

SOLID Concrete
Concrete EC2
Concrete EC2
Cementitious2
Cementitious2 User
Cementitious2 SHCC
Cementitious3
Reinforced Concrete
Microplane M4
SBETA Material
Safety Format
Design
1. The pull down menu with options of predefined materials will appear after the clicking on the arrow. In this case the Concrete EC2 should be chosen.
2. This icon starts the creation of the new material. Once this icon is selected the New SOLID Concrete window appear see Figure 54.
Assign
Draw
Unassign
Exchange
Close

Figure 53: Description of the new material creation

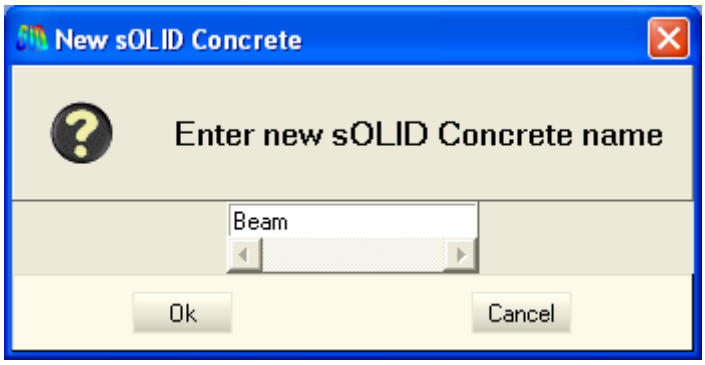


Figure 54: The window for the definition of the New SOLID Concrete

Parameter input:
Enter new SOLID Concrete name:
Beam

When the new material is created, its name will be offered in the pull down menu (see Figure 55). This new material should be selected and then the parameters definition can be changed.

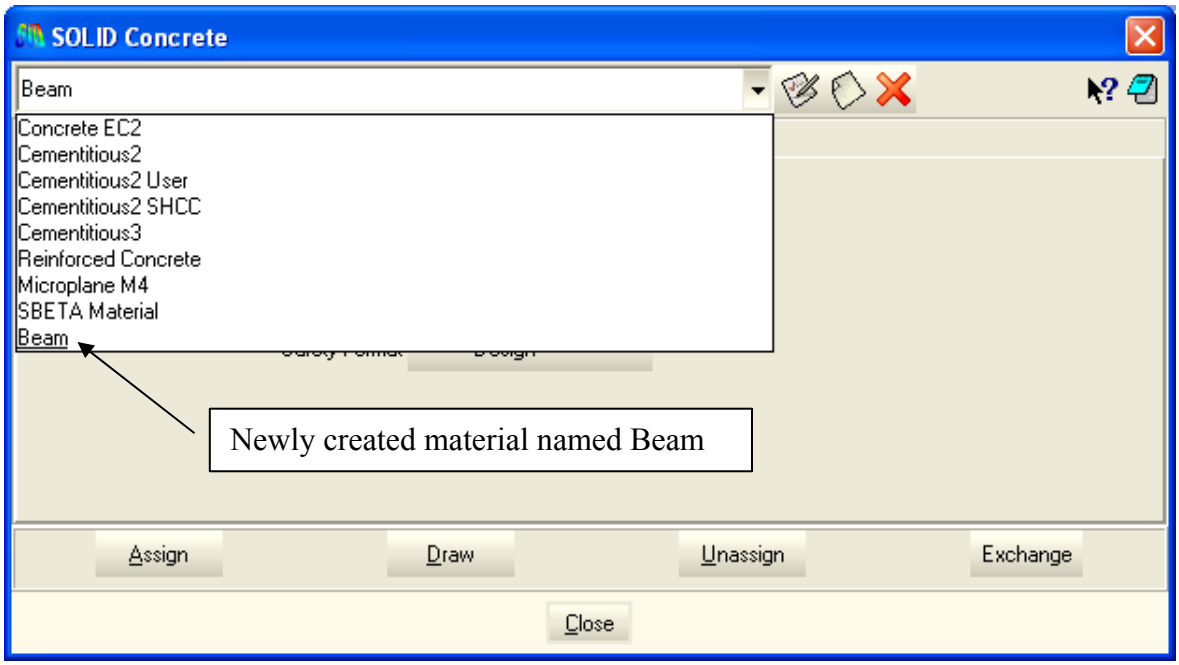


Figure 55: The selection of the New SOLID Concrete material

The parameters of the new material CONCRETE are predefined according to Eurocode 2. In this example it is necessary to have parameters of concrete class **30/37** and Safety Format should be **Mean**. It can be done by selecting this class parameter and safety format in the material window. The process of the class and safety format definition is depicted in the Figure 56. It is very important to select checkbox Generate Material otherwise no parameters will be updated. All parameters definition is completed by clicking on the Update Changes icon .

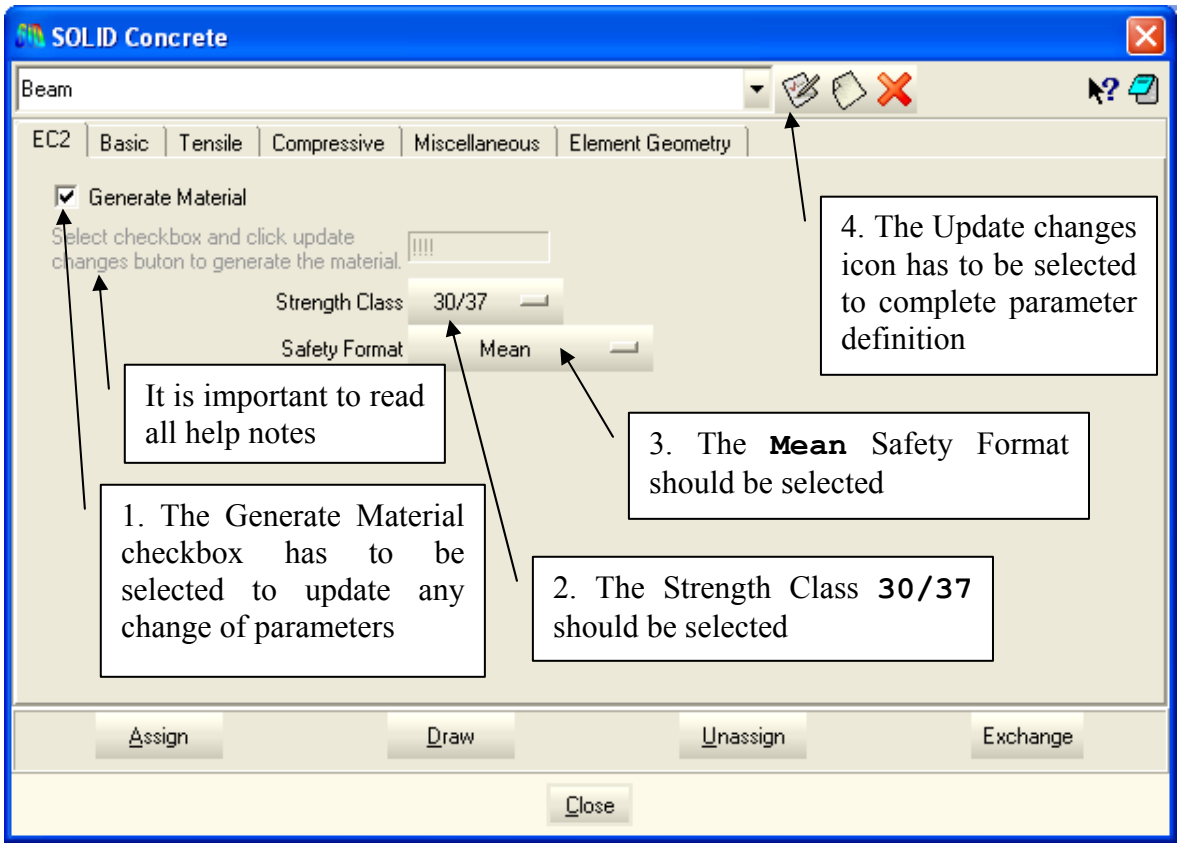


Figure 56: The description of the class definition

After updating of EC 2 parameters, the rest of parameters will change automatically. The following pictures show default parameters of concrete class 30/37.

See Figure 57, Figure 58, Figure 59, Figure 60, and Figure 61.

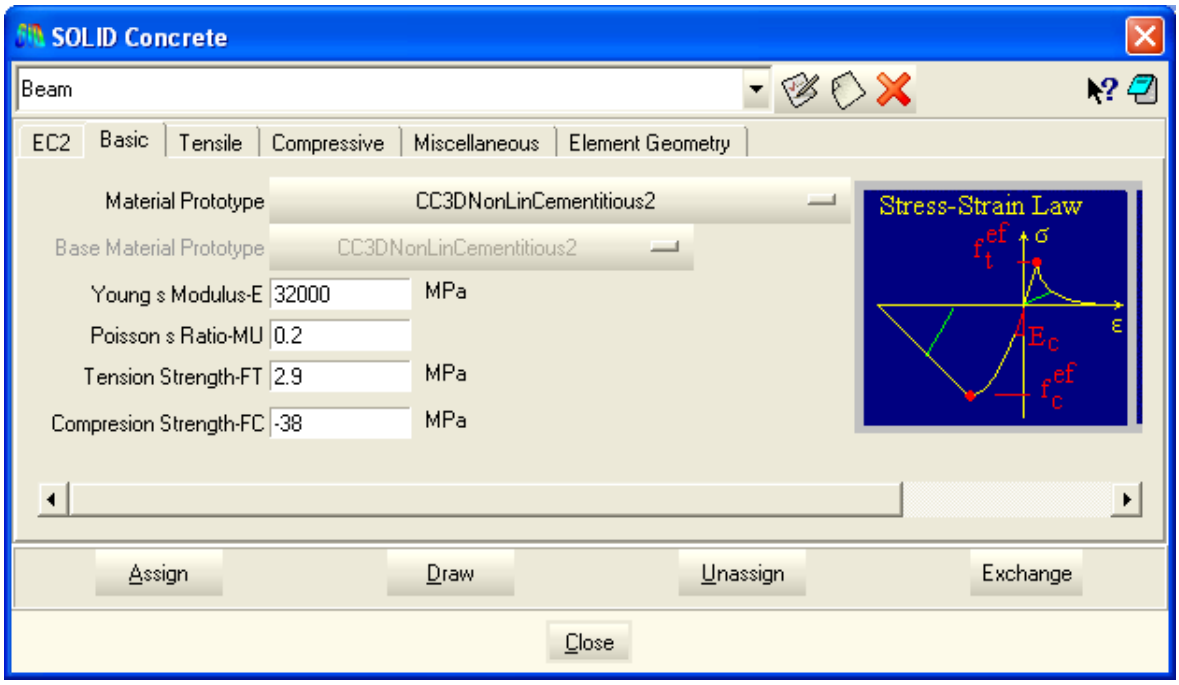


Figure 57: The default Basic parameters of the concrete class 30/37

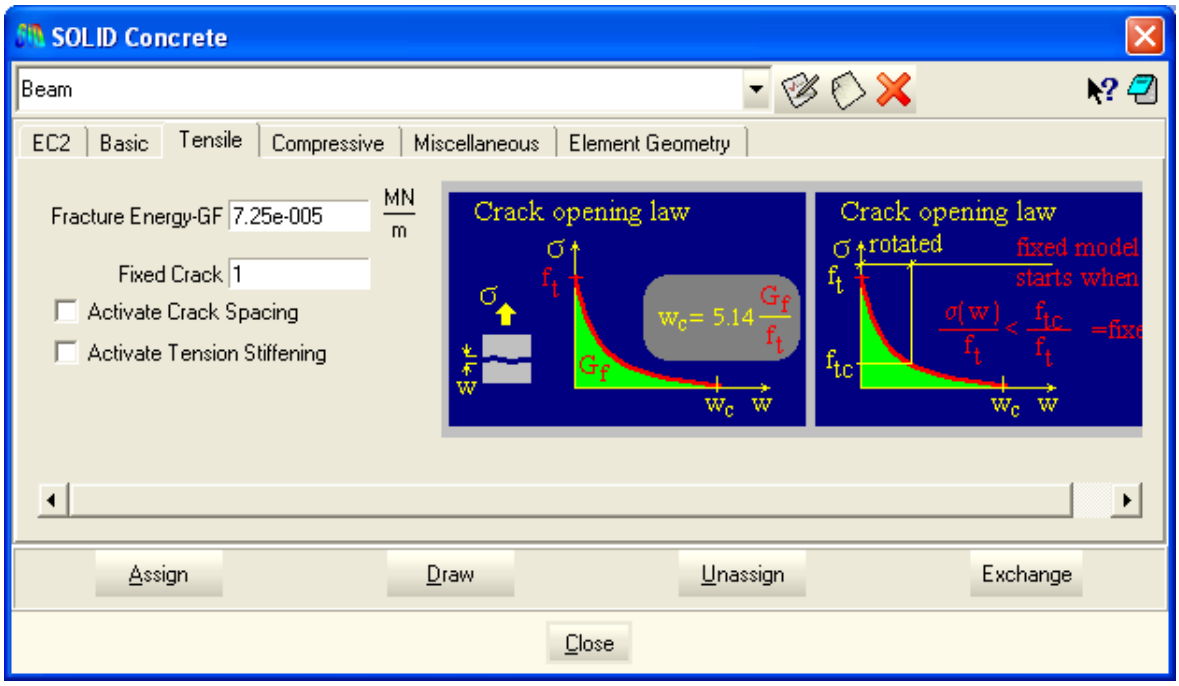


Figure 58: The default Tensile parameters of the concrete class 30/37

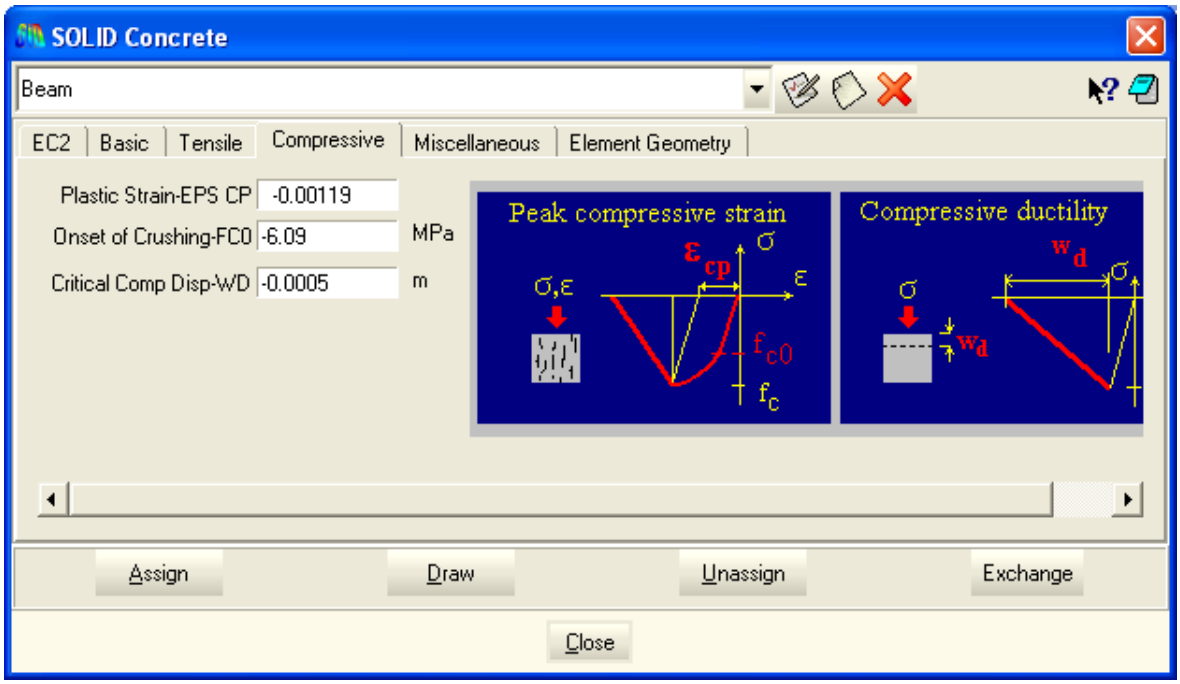


Figure 59: The default Compressive parameters of the concrete class 30/37

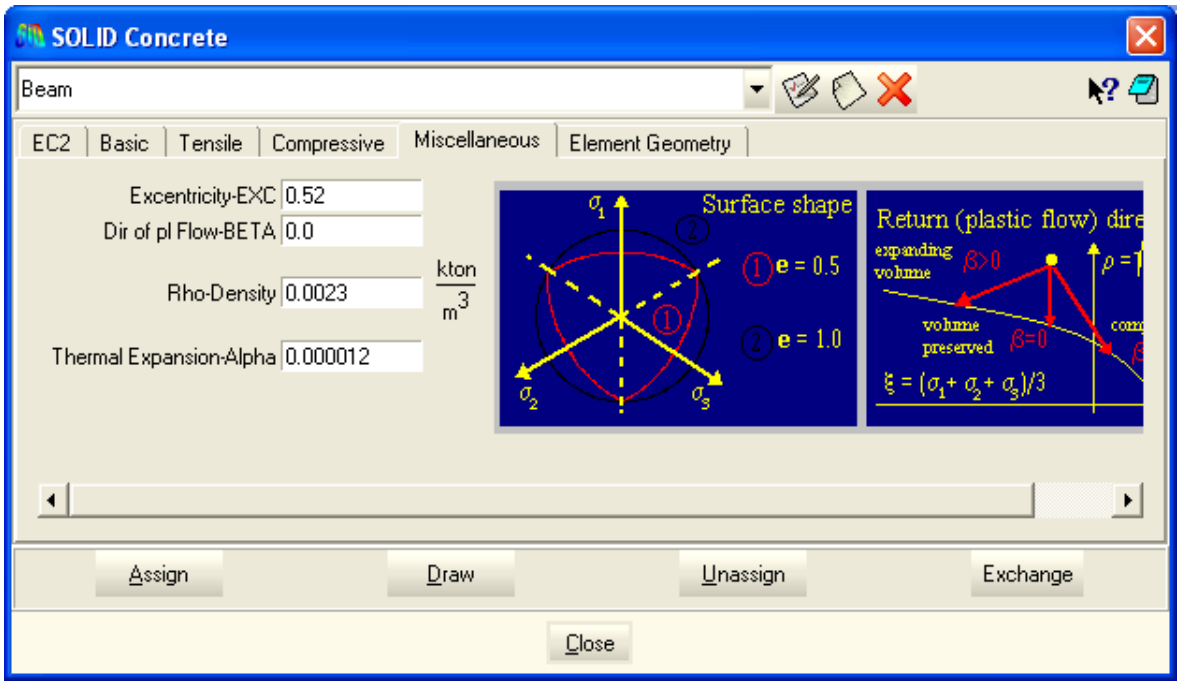


Figure 60: The default Miscellaneous parameters of the concrete class 30/37

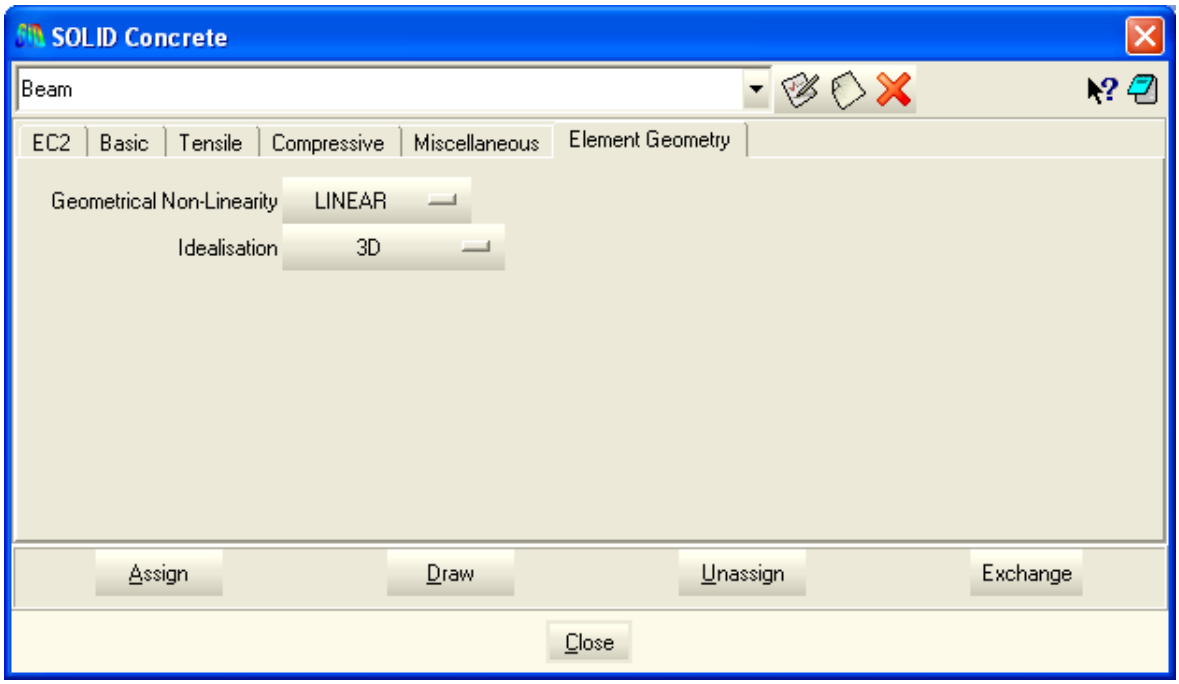


Figure 61: The default Element Geometry parameters of the concrete class 30/37

When the Beam material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **Assign** in the bottom of the material window. After this the several options will appear. In this case the Beam material will be assigned to the beam which is a volume. Therefore the option **Volumes** should be selected. Then the volume of the beam geometry can be selected in the graphical area and the button **Finish** has to be pressed to complete assigning.

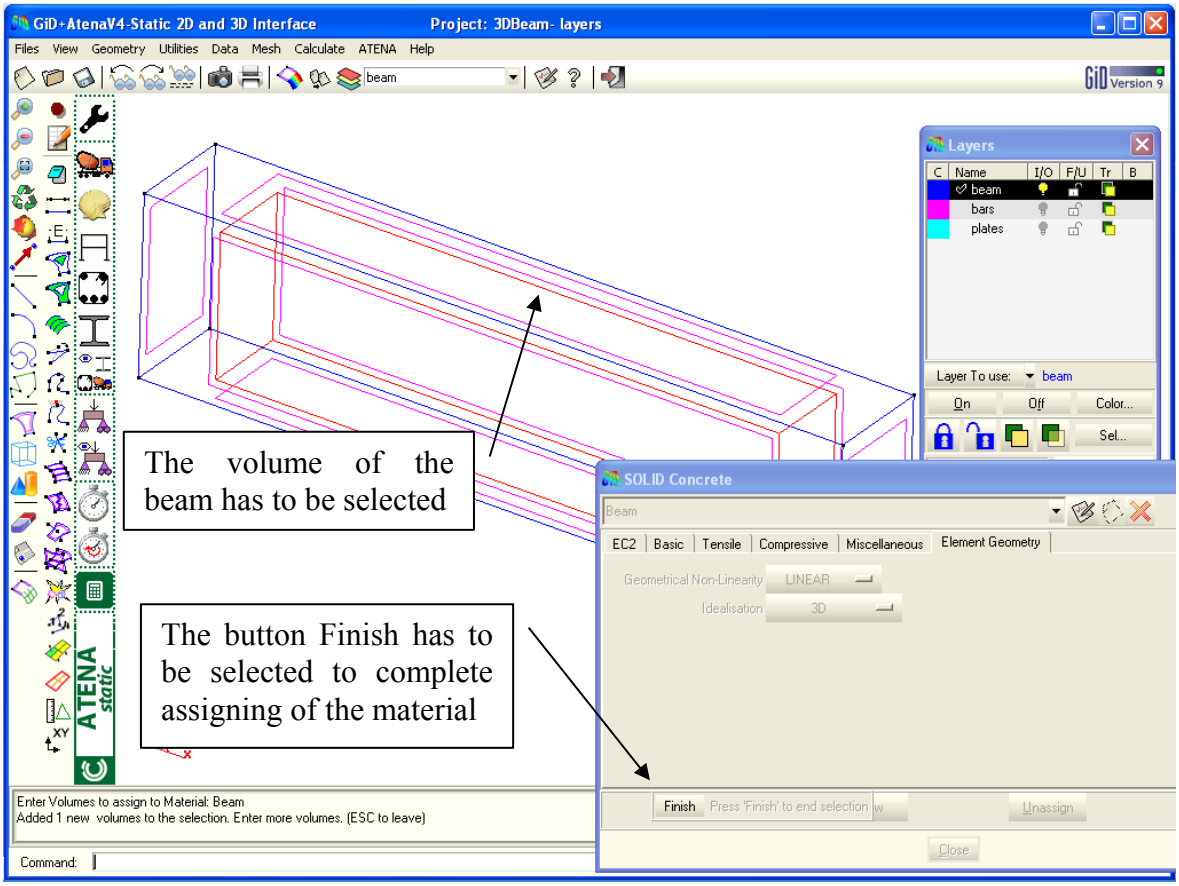

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- layers
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
Layers
C Name I/O F/U Tr B
beam
bars
plates
Layer To use: beam
On Off Color...
Sel...
The volume of the beam has to be selected
The button Finish has to be selected to complete assigning of the material
SOLID Concrete
Beam
EC2 Basic Tensile Compressive Miscellaneous Element Geometry
Geometrical Non-Linearity LINEAR
Idealisation 3D
Finish Press 'Finish' to end selection
Unassign
Close
Enter Volumes to assign to Material: Beam
Added 1 new volumes to the selection. Enter more volumes. (ESC to leave)
Command:
ATENA static

Figure 62: The assigning of the CONCRETE material to the volume

The beam material was created and assigned. Now, in the following section the steel plates material can be created.

3.3.2 Loading and supporting steel plates

Before definition of the loading and supporting plates material it is good to display only the plates layer.

Loading and supporting steel plates are made from elastic material. The material definition of plates starts with the command **`Data | Materials | SOLID Elastic`** in the main menu (see Figure 63).

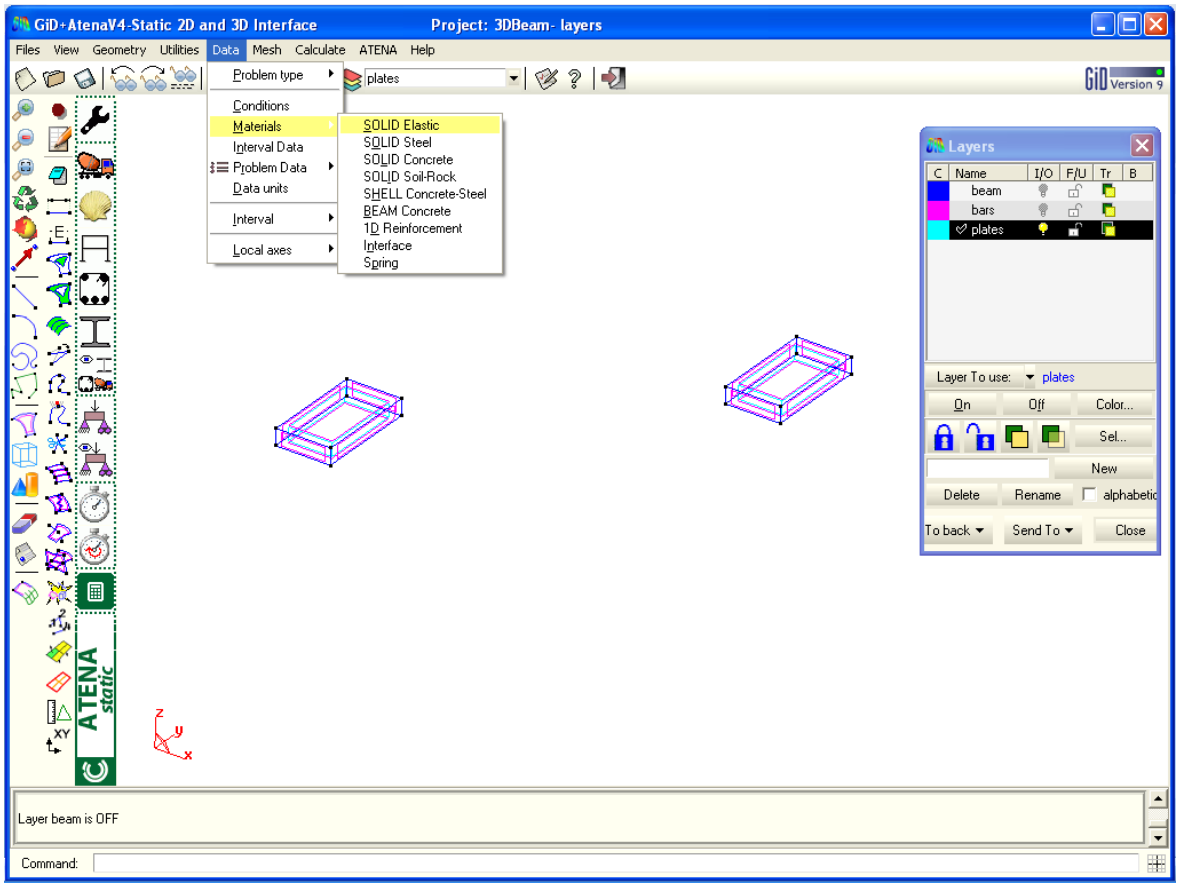


Figure 63: The selection of the command for the definition of the plates material

After the selection of this command the window for the definition of the SOLID Elastic will appear (seeFigure 64).

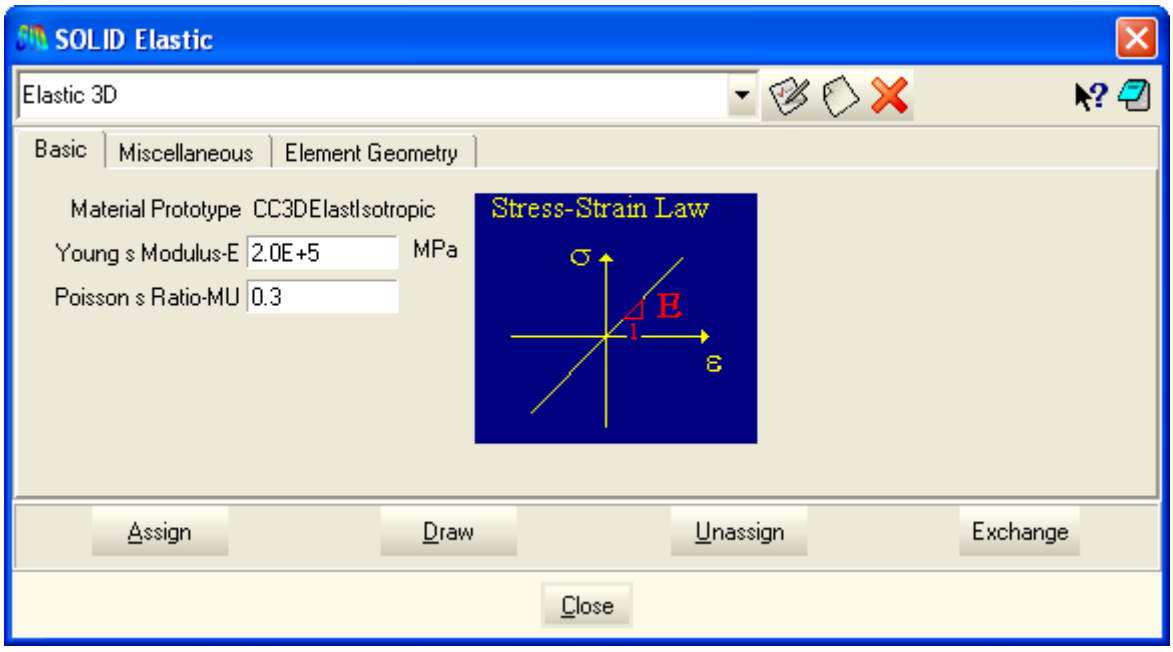


Figure 64: The window for the definition of the SOLID Elastic

The process of the Elastic material creation is very similar to the creation of the Concrete material. First, it is important to copy material definition of the already existing material and save it under the new name. There is only one elastic material and it will be chosen to be copied for the material of this example. The Elastic 3D should be selected.

After the selection of the predefined material the icon New SOLID Elastic should be pressed. The selection of this material and selection of the New SOLID Elastic icon are depicted in the Figure 65.

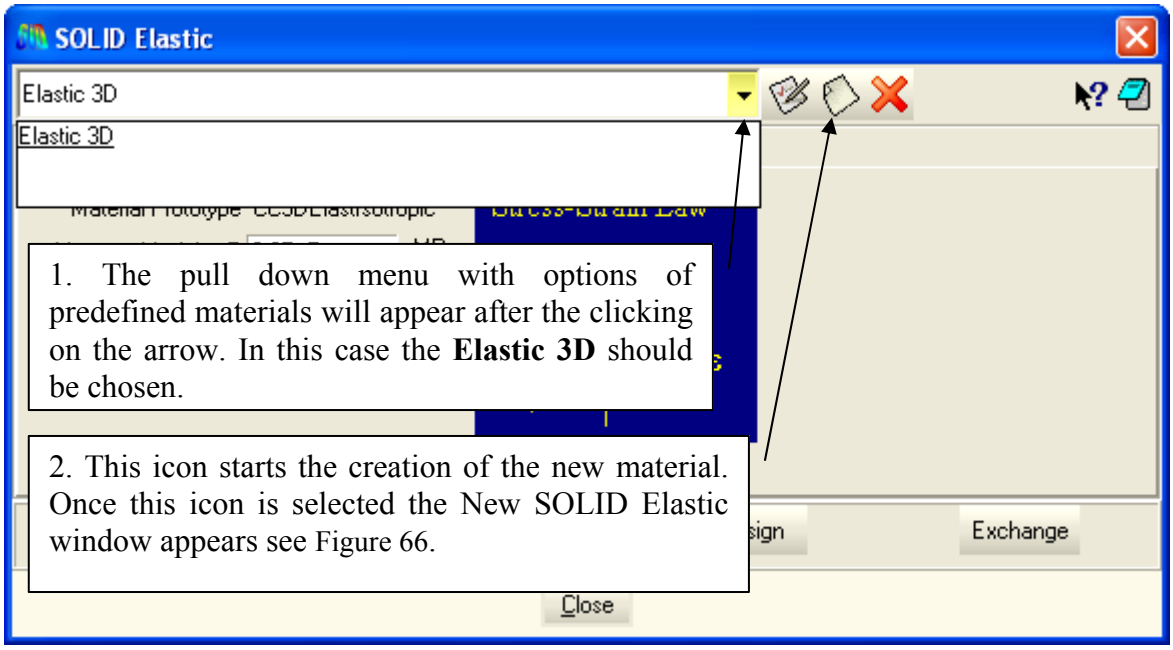

SOLID Elastic
Elastic 3D
Elastic 3D
1. The pull down menu with options of predefined materials will appear after the clicking on the arrow. In this case the **Elastic 3D** should be chosen.
2. This icon starts the creation of the new material. Once this icon is selected the New SOLID Elastic window appears see Figure 66.
Exchange
Close

Figure 65: Description of the new elastic material creation

After the selection of the icon New SOLID Elastic the new window for the definition of the new material name will appear (see Figure 66). There the **Plates** name should be written and then it is necessary to press **OK** button to complete this command.

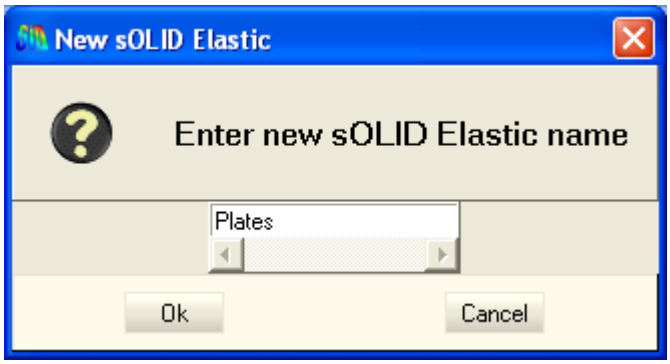


Figure 66: The window for the definition of the New SOLID Elastic material

Parameter input:
Enter new SOLID Elastic name:
Plates

Then the new material should be selected and then the parameter definition can be changed by clicking on the icon . In the case of the elastic material the default parameters will be left unchanged.

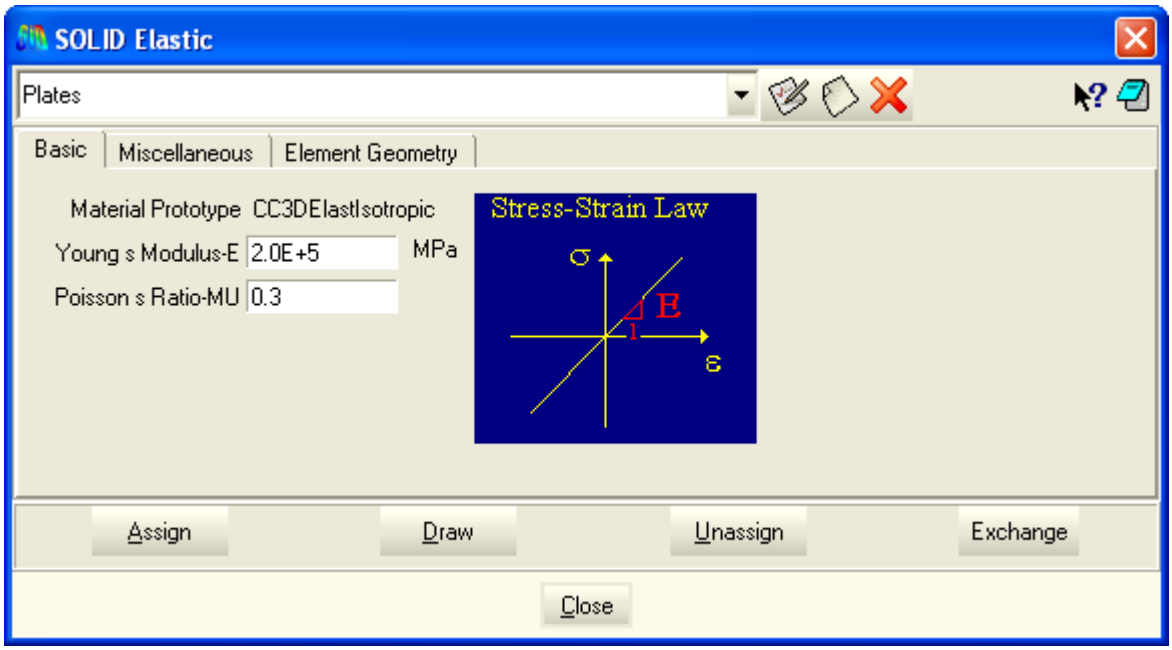


Figure 67: The default Basic parameters of the elastic material

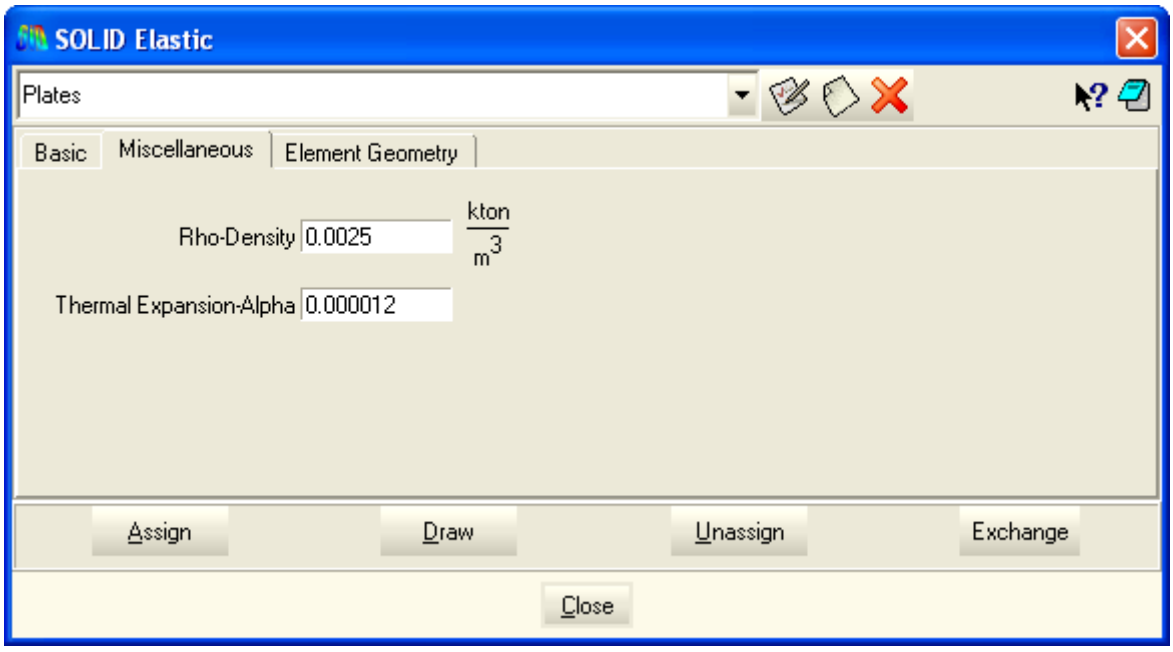


Figure 68: The default Miscellaneous parameters of the elastic material

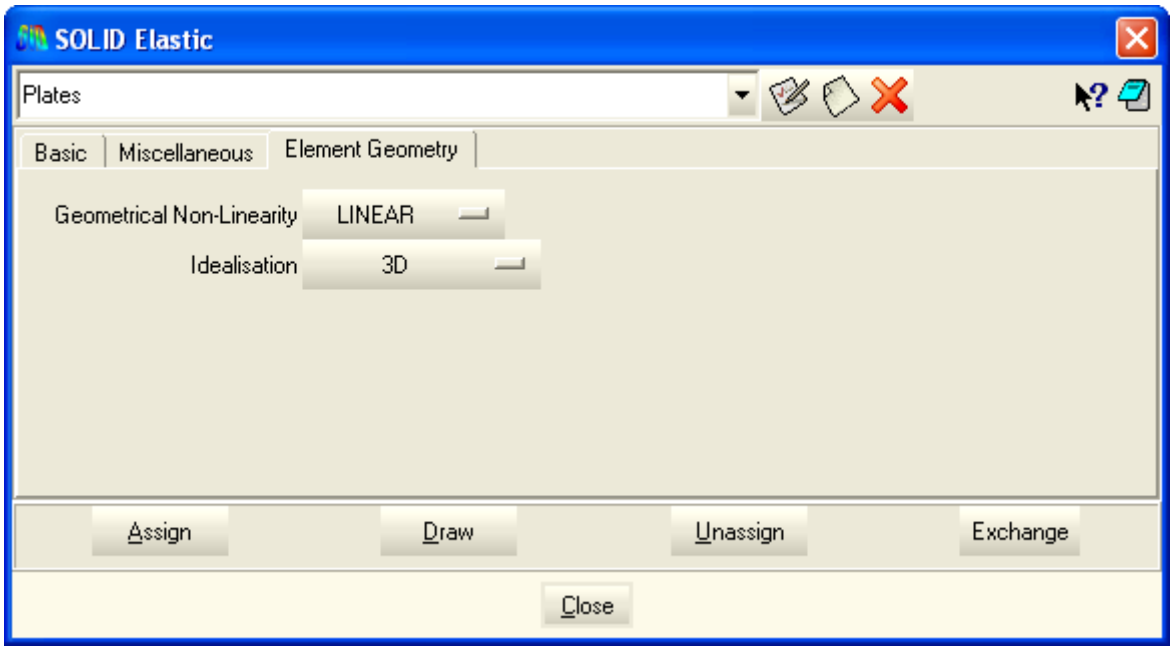


Figure 69: The default Element Geometry parameters of the elastic material

When the elastic material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **`Assign`** in the bottom of the material window. After selecting this button the several options will appear. In this case the **Plates** material will be assigned to the loading and supporting steel plates which are volumes. Therefore the option **`Volumes`** should be selected. Then the volumes of the plates can be selected in the graphical area and the button **`Finish`** has to be pressed to complete assigning.

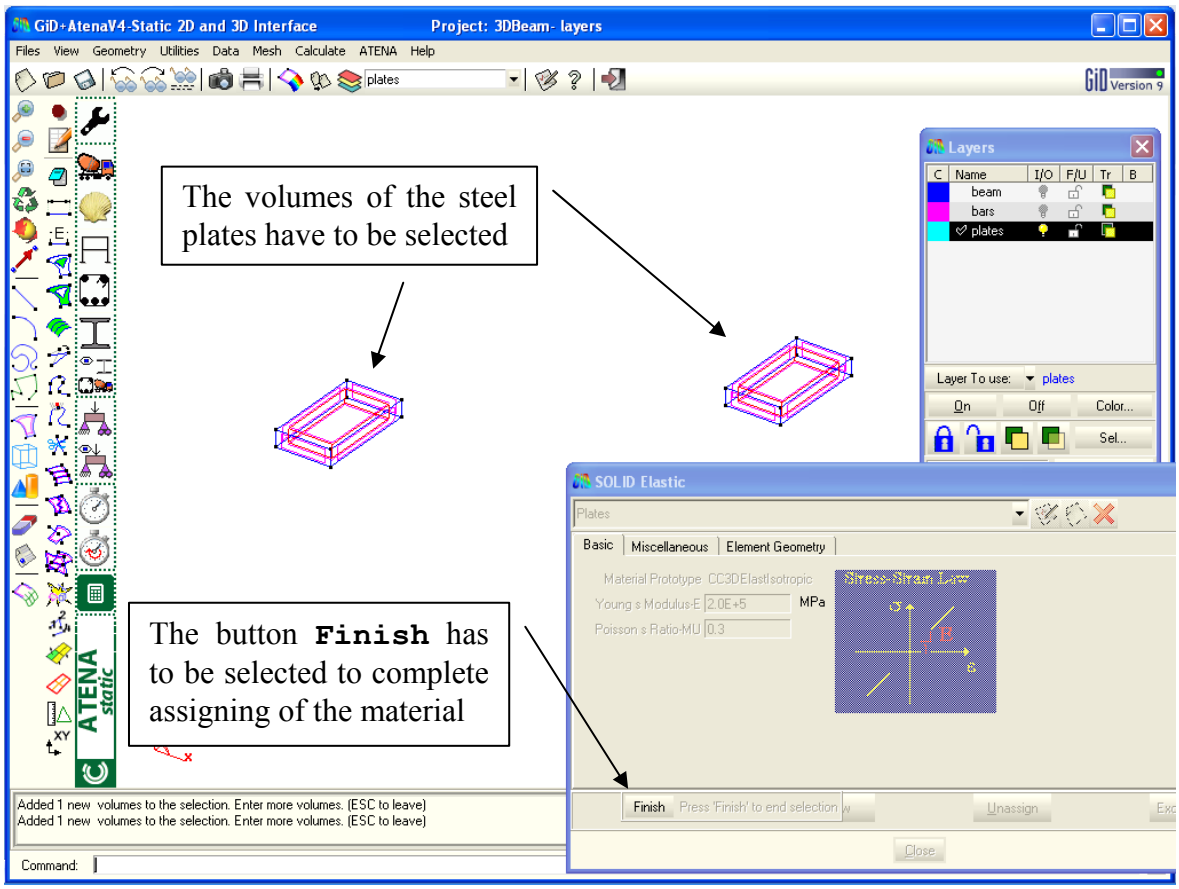


Figure 70: The assigning of the Plates material to the volumes

The steel plates material was created and assigned. In the last section the reinforcement material will be created.

3.3.3 Reinforcement bars

Before definition of the reinforcement material it is good to display only the Bars layer.

The material definition of the reinforcement starts by the selecting the icon or with the command **`Data | Materials | 1D Reinforcement`** (see Figure 71).

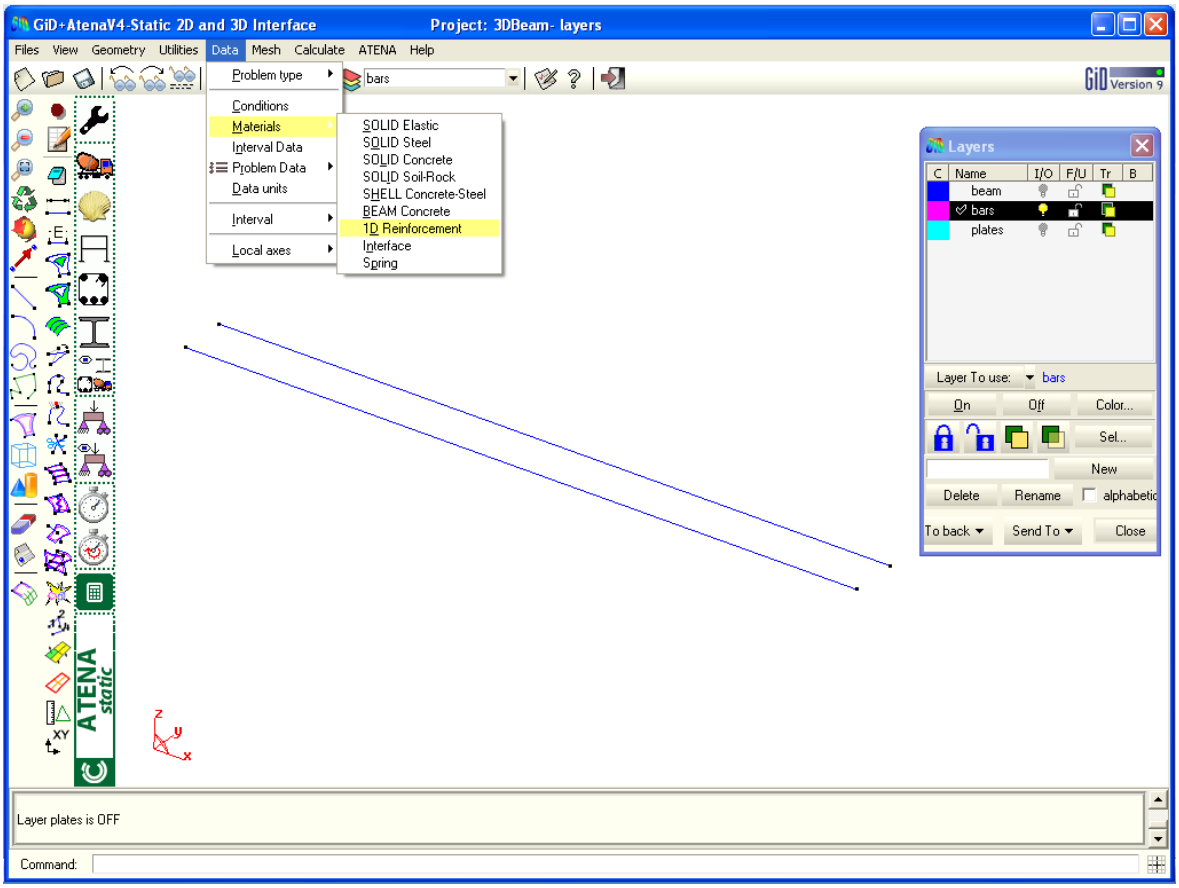


Figure 71: The selection of the command for the definition of the reinforcement material

After the selection of this command the window for the definition of the 1D Reinforcement will appear (see Figure 72).

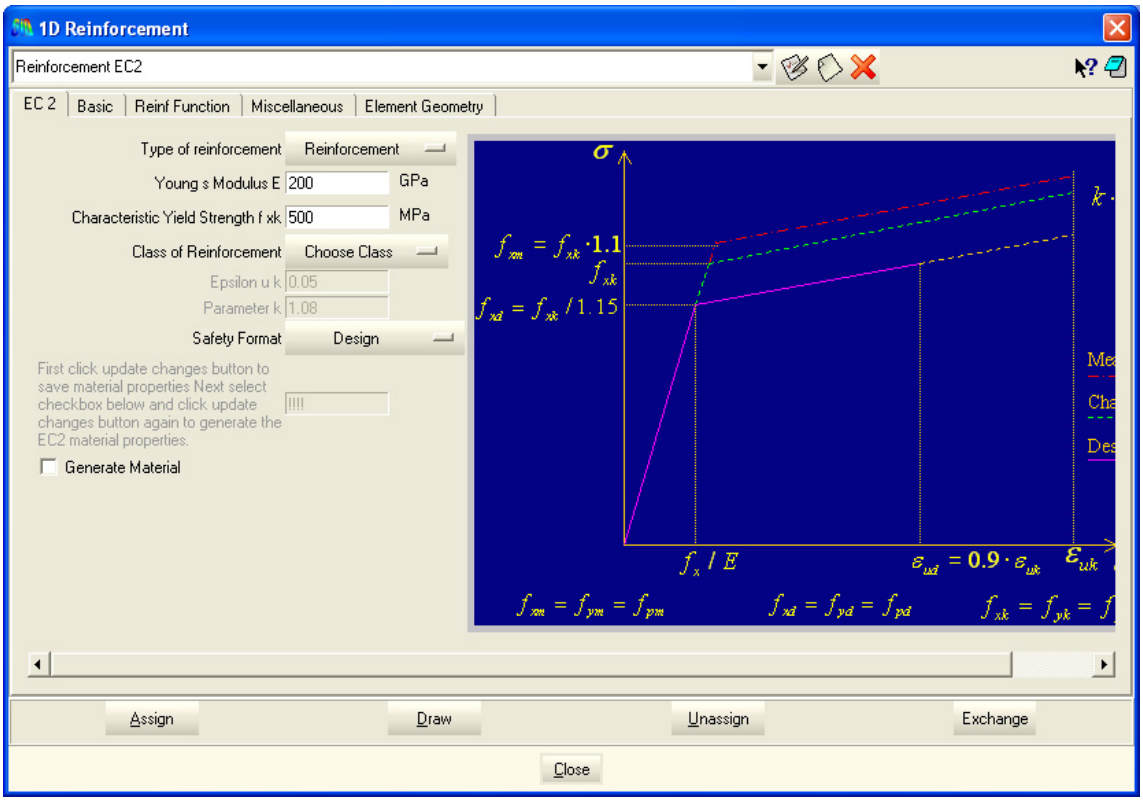


Figure 72:The window for the definition of the 1D Reinforcement

First, it is important to copy material definition of the already existing material and save it under the new name. In this case the new name will be **Bars**. The predefined material Reinforcement EC2 should be chosen for the copying. After the selection of the predefined material the icon New 1D Reinforcement should be selected. After the selection of the New 1D Reinforcement icon the new window for the definition of the new material name will appear (see Figure 73). There the **Bars** name should be written and then it is necessary to press **OK** button to complete this command.

Figure 73:The window for the definition of the New 1D Reinforcement

Parameter input:
Enter new 1D Reinforcement name:
Bars

This new material should be selected and then can be changed the parameter definition. The parameters of the new material **Bars** are predefined according to Eurocode 2. In this example the **Characteristic Yield Strength** should be **560 MPa** and **Class of Reinforcement** should be **A**. The parameters definition is depicted in the Figure 74. It is very important to select checkbox Generate Material otherwise no parameters will be updated. All parameters definition is completed by clicking on the Update Changes icon .

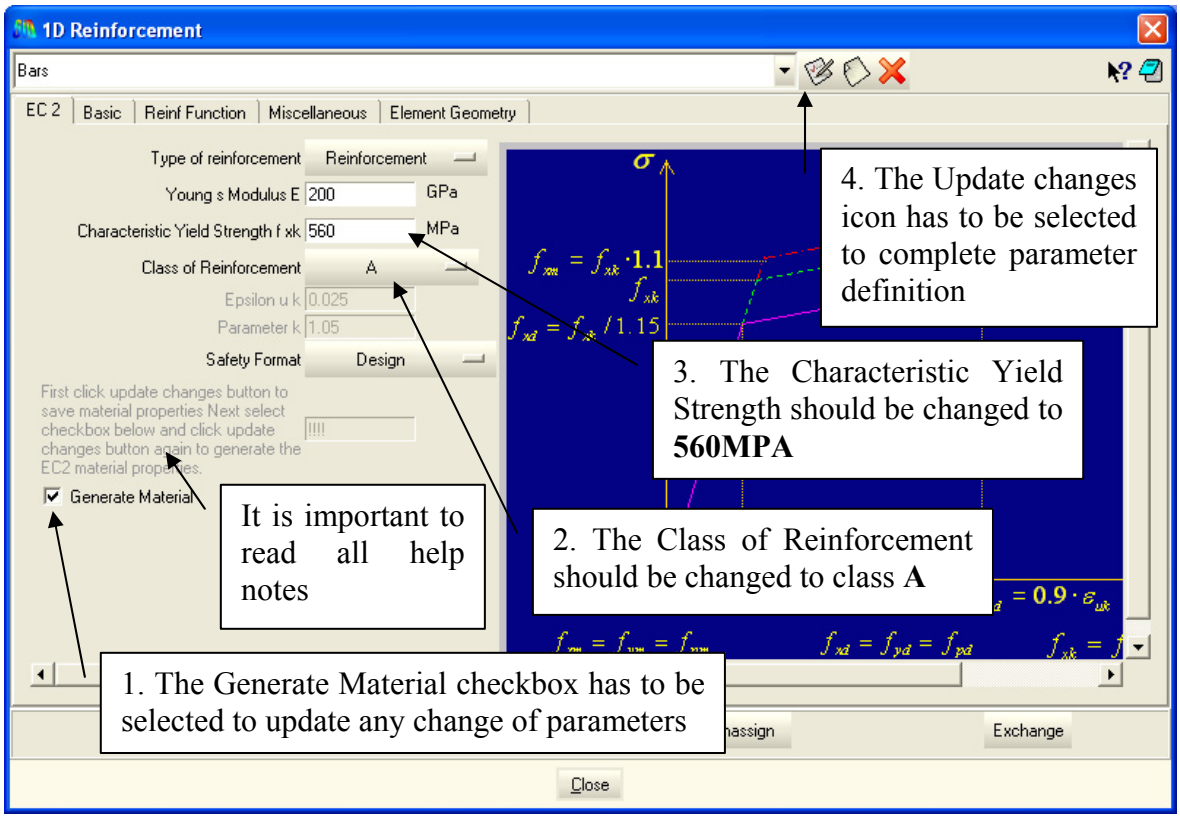


Figure 74: The description of the reinforcement definition

In the Basic properties there is a bars definition. By the checking the checkbox Calculator the windows for the profile definition will appear. The Profile should be 26 mm and number of profiles will stay 1. Then the Update changes icon has to be clicked to recalculate reinforcement area. Then it is necessary to click on the Update changes icon again to save all changes into the material (see Figure 75).

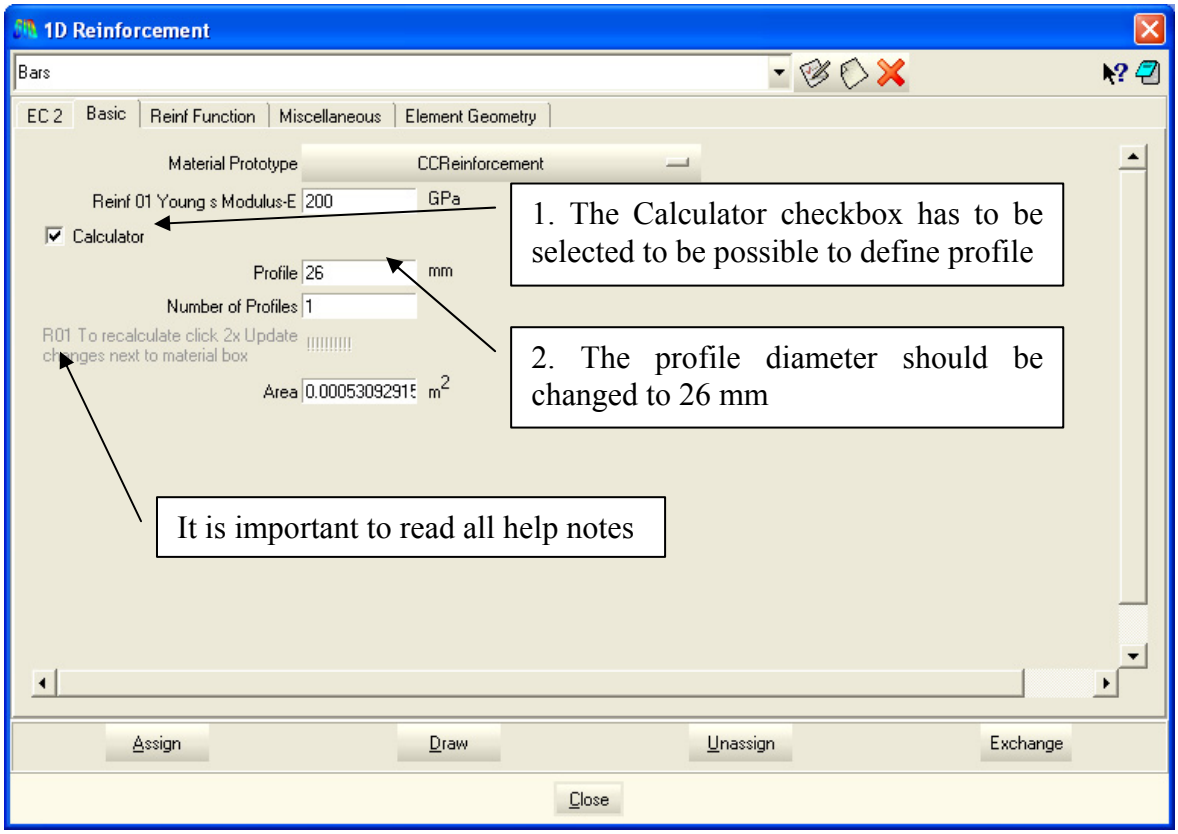


Figure 75: The default Basic parameters of the reinforcement, the icon Update changes has to be clicked 2x to change parameters

The rest of the reinforcement parameters will be default. There is no change necessary (see Figure 76, Figure 77, and Figure 78).

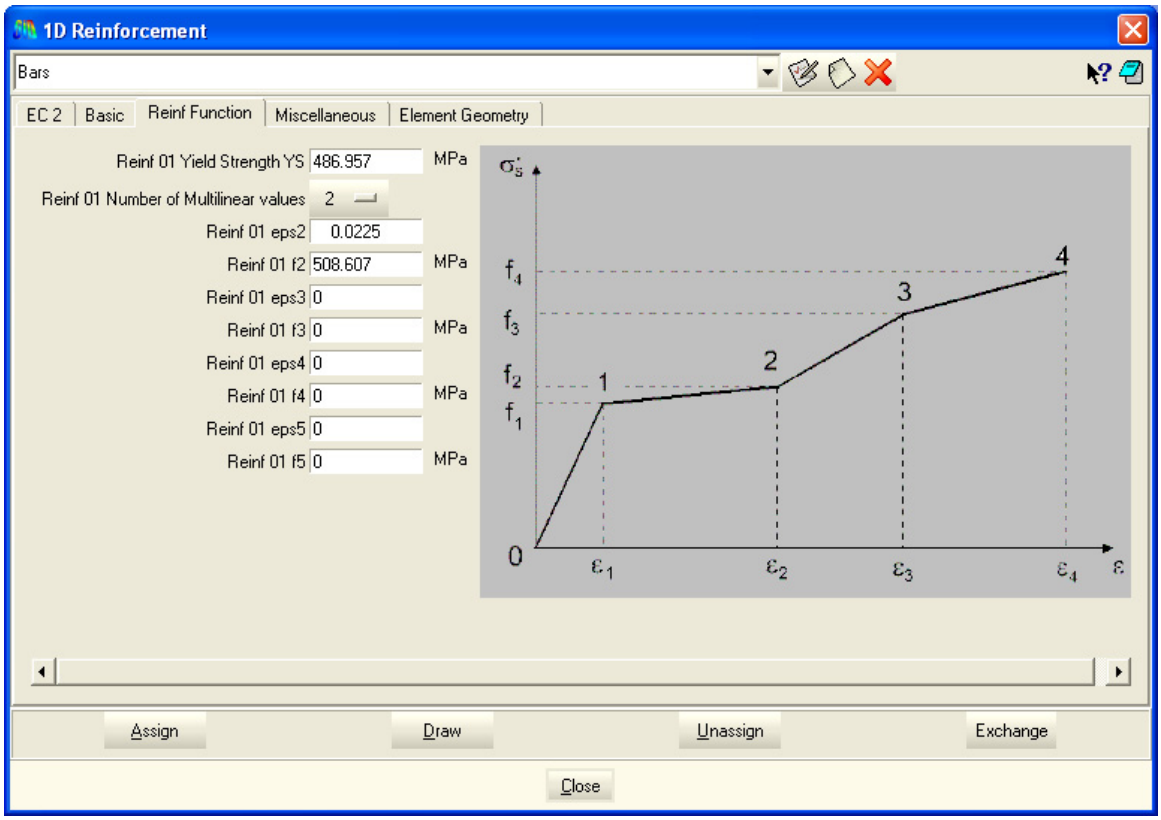


Figure 76: The default Reinf Function parameters of the reinforcement

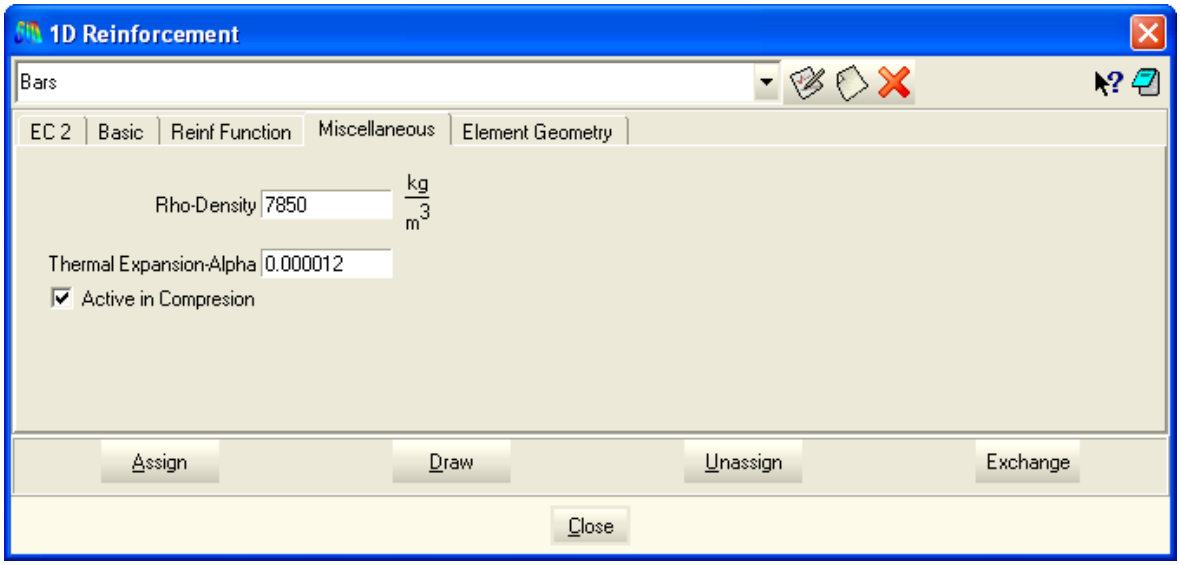


Figure 77: The default Miscellaneous parameters of the reinforcement

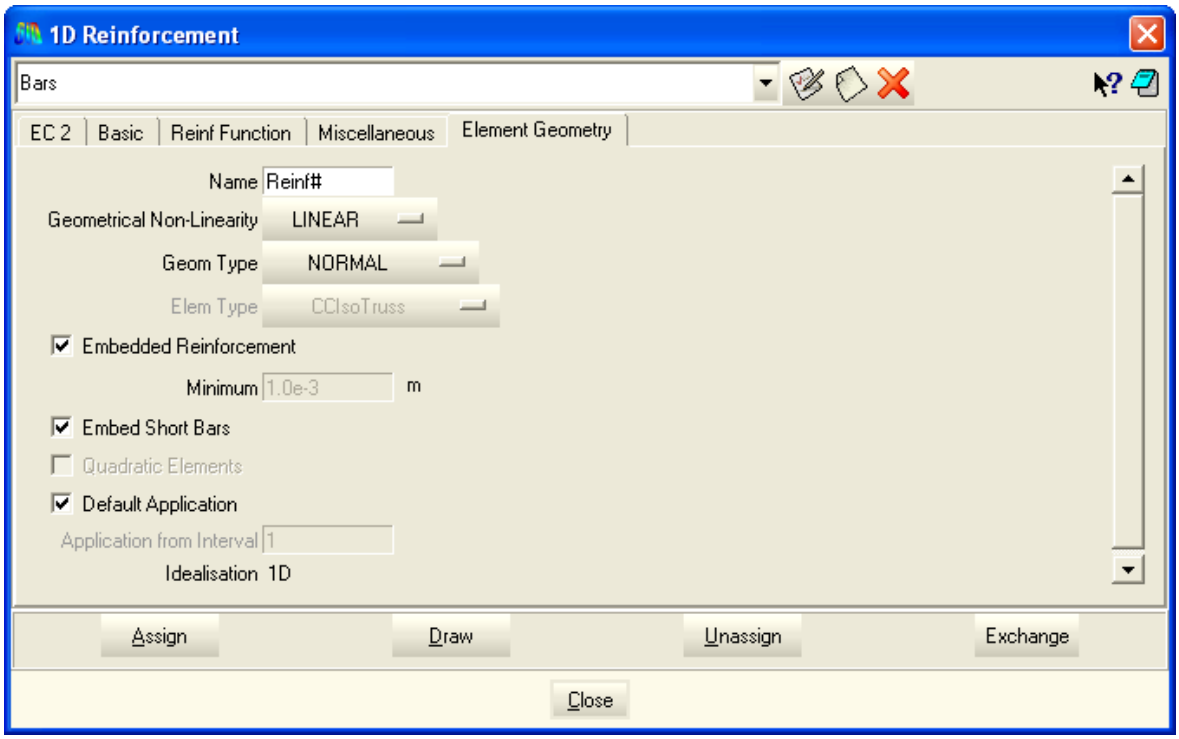


Figure 78: The default Element Geometry parameters of the reinforcement

When the bars material parameters are defined the material can be assigned to the geometry. It is done by selecting the button **Assign** in the bottom of the material window. After this the several options will appear. In this case the Bars material will be assigned to two straight lines. Therefore the option Lines should be selected. Then the lines of the reinforcement can be selected in the graphical area and the button **Finish** has to be pressed to complete assigning (see Figure 79).

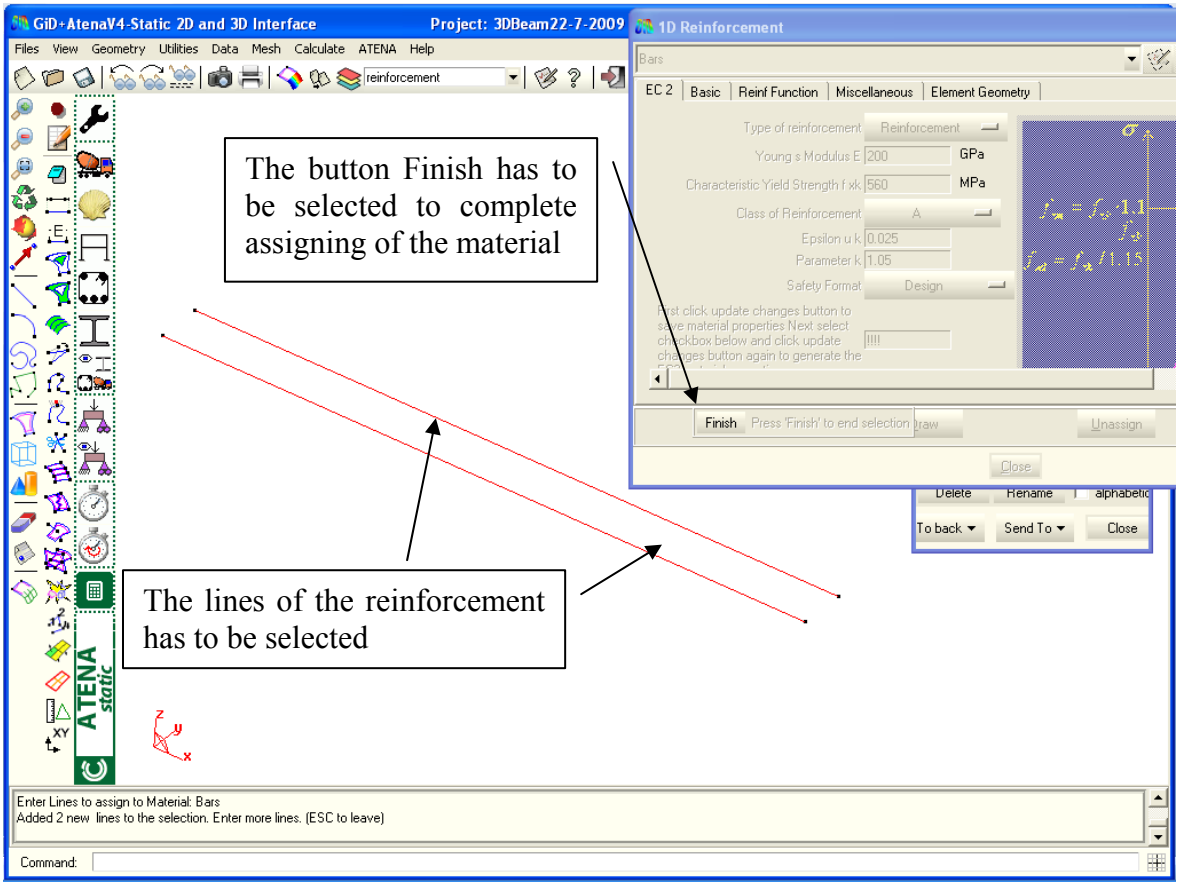

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam22-7-2009
1D Reinforcement
The button Finish has to be selected to complete assigning of the material
The lines of the reinforcement has to be selected
Finish
Unassign
Close
Enter Lines to assign to Material: Bars
Added 2 new lines to the selection. Enter more lines. (ESC to leave)
Command:

Figure 79: The assigning of the Bars material into lines

All materials are created and assigned. The icon Draw all materials can be used to control if all materials are well assigned. But before that it is important to display all layers and their content. It is simply done by clicking on the grey bulb which should change to the yellow colour after the clicking. Then the Draw all materials icon can be used. See Figure 80.

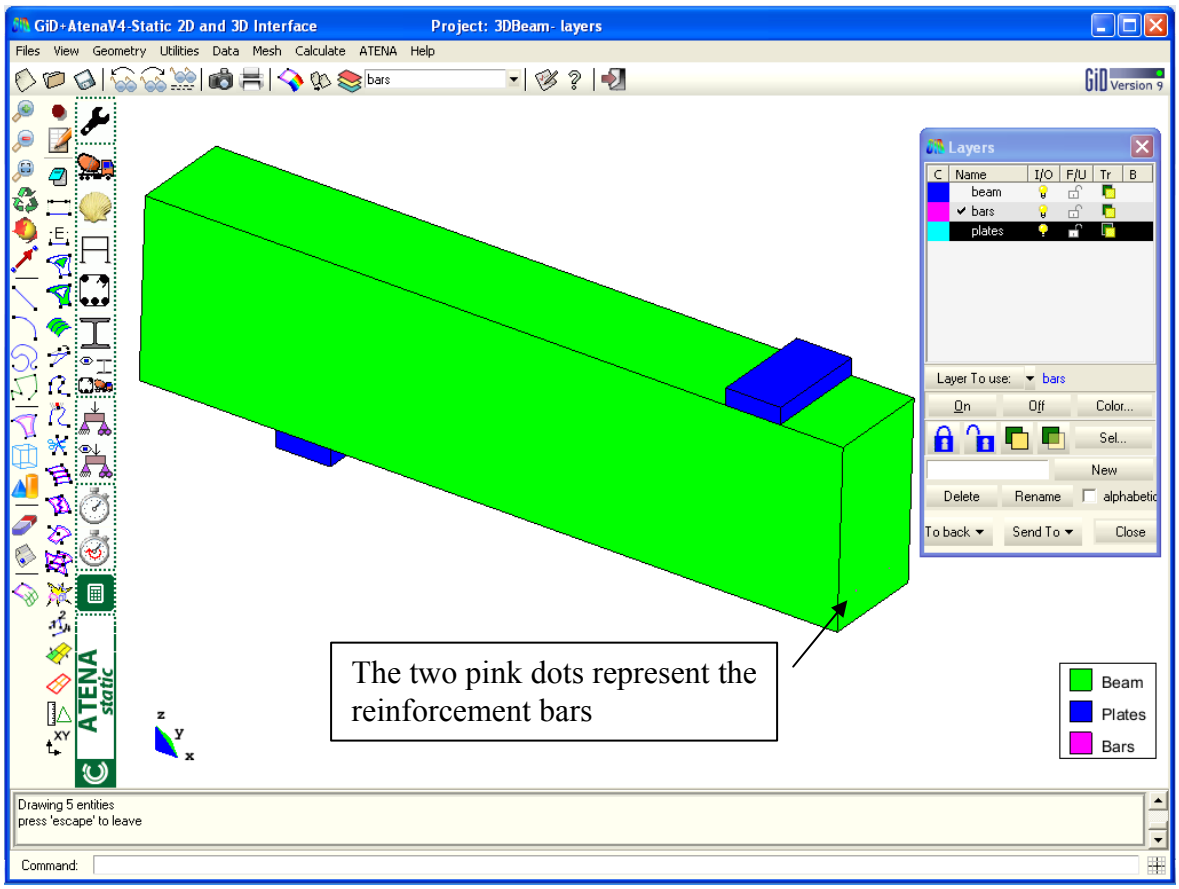


Figure 80: The drawn assigned materials

3.4 Boundary conditions

In this chapter the boundary condition are described. The analyzed beam is supported at the bottom steel plate in the vertical direction. There the support condition will be defined. Since only a symmetric half of the beam is analyzing, it is necessary to enforce the fixed condition along the right side of the beam. It means that the horizontal displacements along x-axis should be equal zero.

The beam is loaded at the top steel plate. The object of this example is to determine the maximal load-carrying capacity of the beam. It means that it should be possible to trace structural response also in the post-peak regime. The easiest method to accomplish this is by loading the beam by prescribed displacements condition at the top steel plate.

It is important to monitor forces, displacement or stresses during non-linear analysis. The monitor data are important information about the state of the structure. For instant from monitoring of applied forces, it is possible to determine if the maximal load was reached or not.

The overall of the conditions there are four types of the boundary conditions in this example – monitors, support, displacement and symmetry conditions.

3.4.1 Support

The analyzed beam is supported at the bottom steel plate in the vertical direction. The support condition should be applied to the line. This line has to be added into the bottom plate geometry. It will be done by dividing the bottom plate surface.

The steel plates are placed in the plates layer. Therefore the plate layer should be activated and displayed. The bars layer can be hidden but the beam layer is better to keep displayed to be able recognize the bottom surface. It is also recommended to zoom in the bottom plate. Make sure that the zoomed surface is the bottom surface of the bottom plate (see Figure 81 and Figure 82).

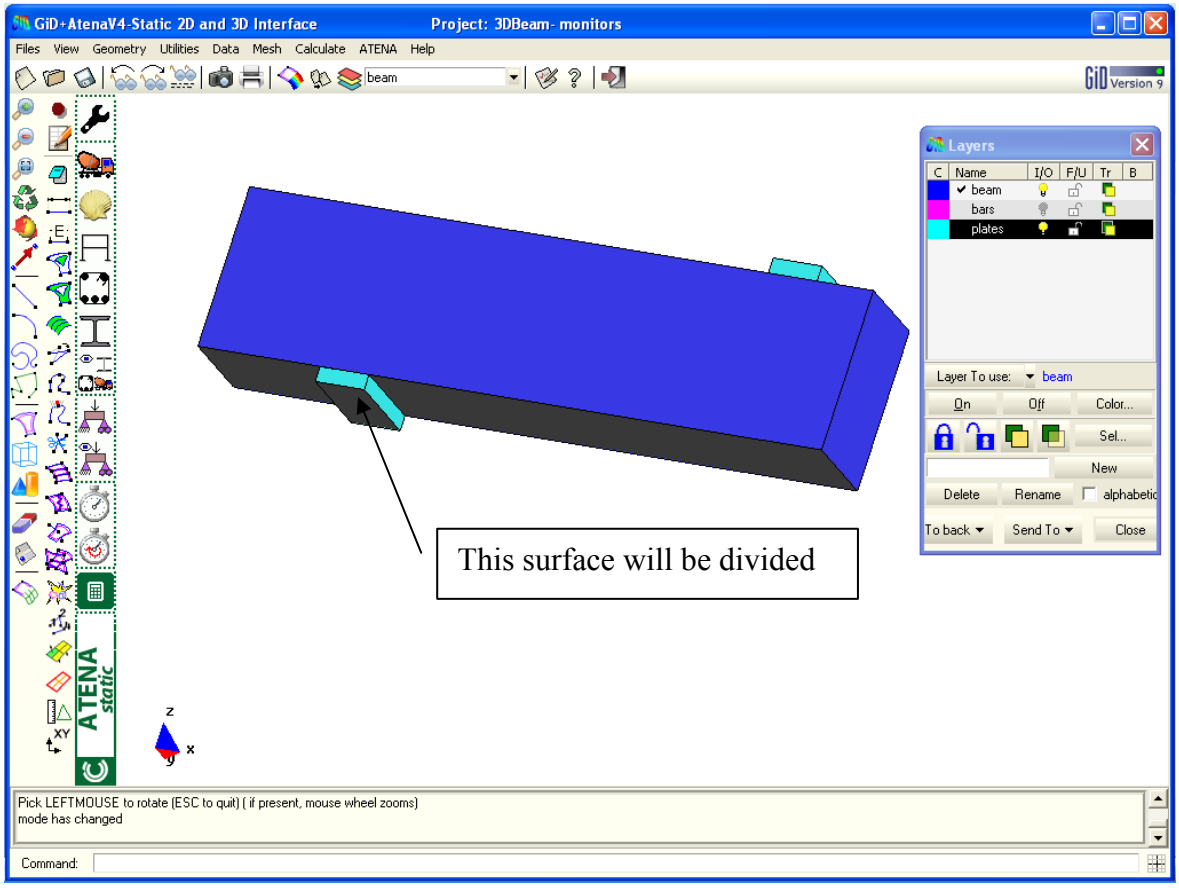


Figure 81: The bottom surface of the bottom plate

The division of the surface starts with executing of the command from main menu **Geometry | Edit | Divide| Surfaces | Num Division** or by selecting of the Divide surface icon (see Figure 82).

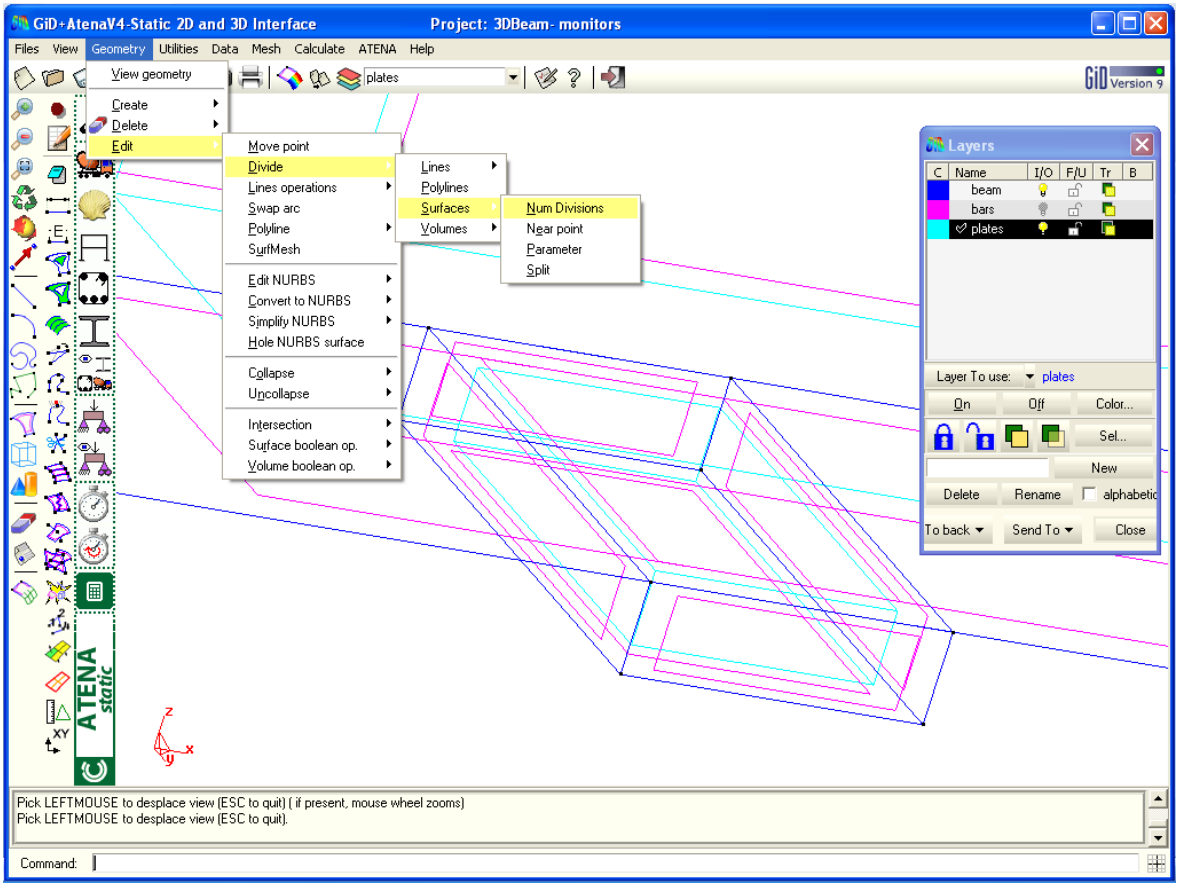


Figure 82: The executing of the division command

After the executing the divide command the cursor will change into this shape and the surface required for dividing should be selected. Once the surface is selected the dialog window will appear on the screen (see Figure 83).

This dialog window asks in which direction the surface should be divided. There are U and V direction and in the graphical area it is possible to see green axis representing U and V direction. In this case USense should be chosen. Once the USense button is chosen the program asks for the number of the divisions. Bottom surface should be divided into two parts (see Figure 84).

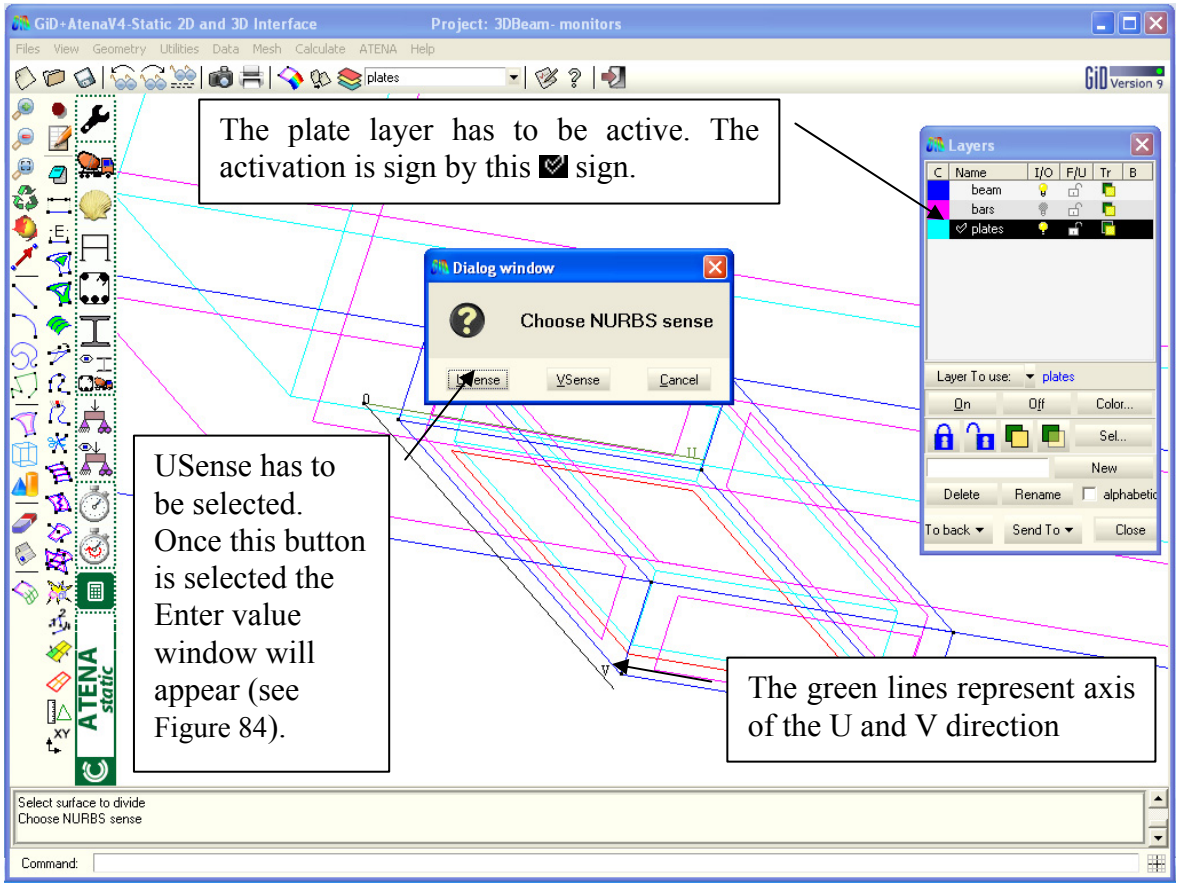


Figure 83: The dividing of the surface

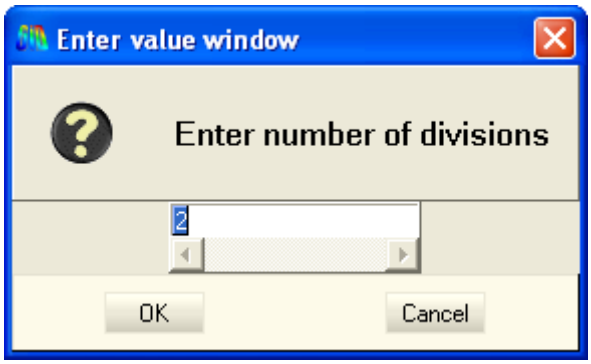


Figure 84: The enter value window

Parameter input:
Enter number of divisions: 2

The button **OK** should be pressed on the enter value window. After that the surfaces is divided (see Figure 85).

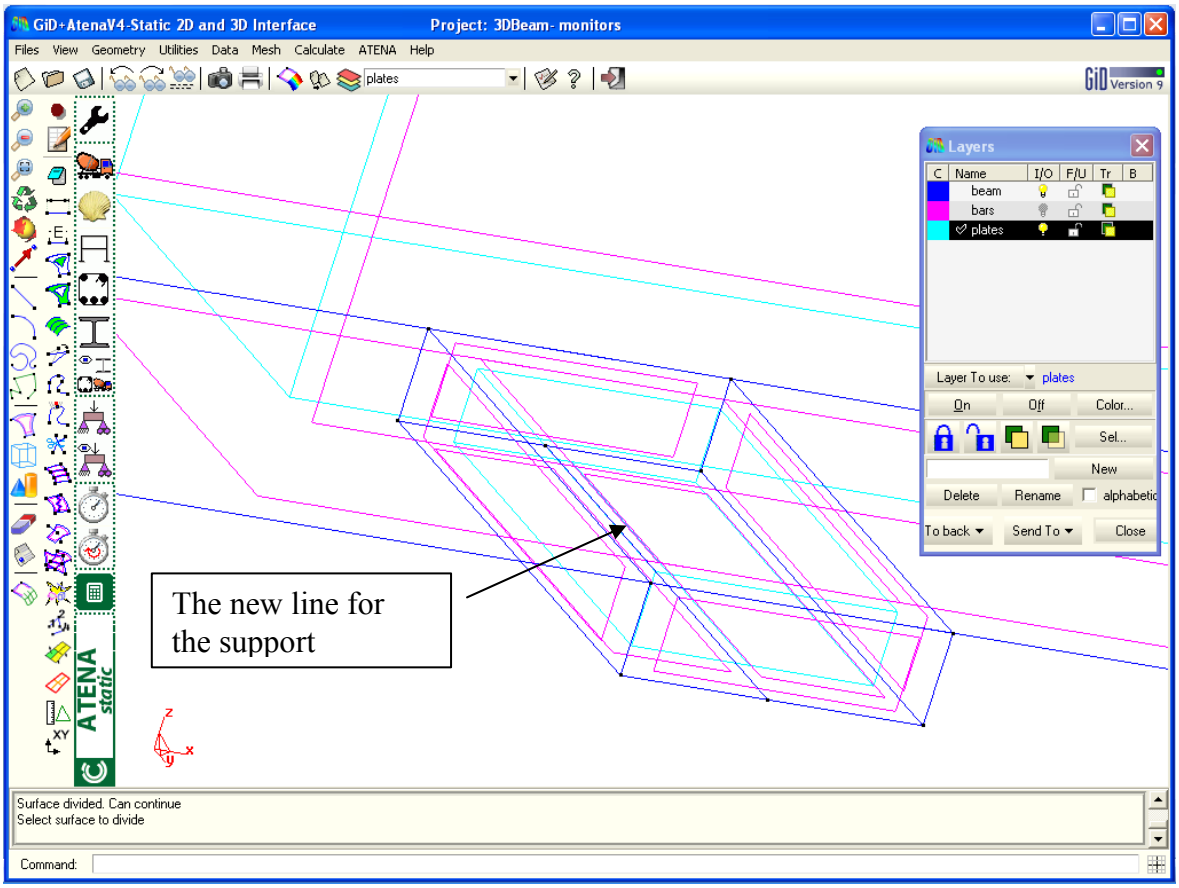


Figure 85: The divided top surface

When the geometry for the support is created the monitor condition can be defined. Conditions command can be executed by the **Data | Conditions** in the main menu or by the icon . The support condition definition is depicted in the Figure 86.

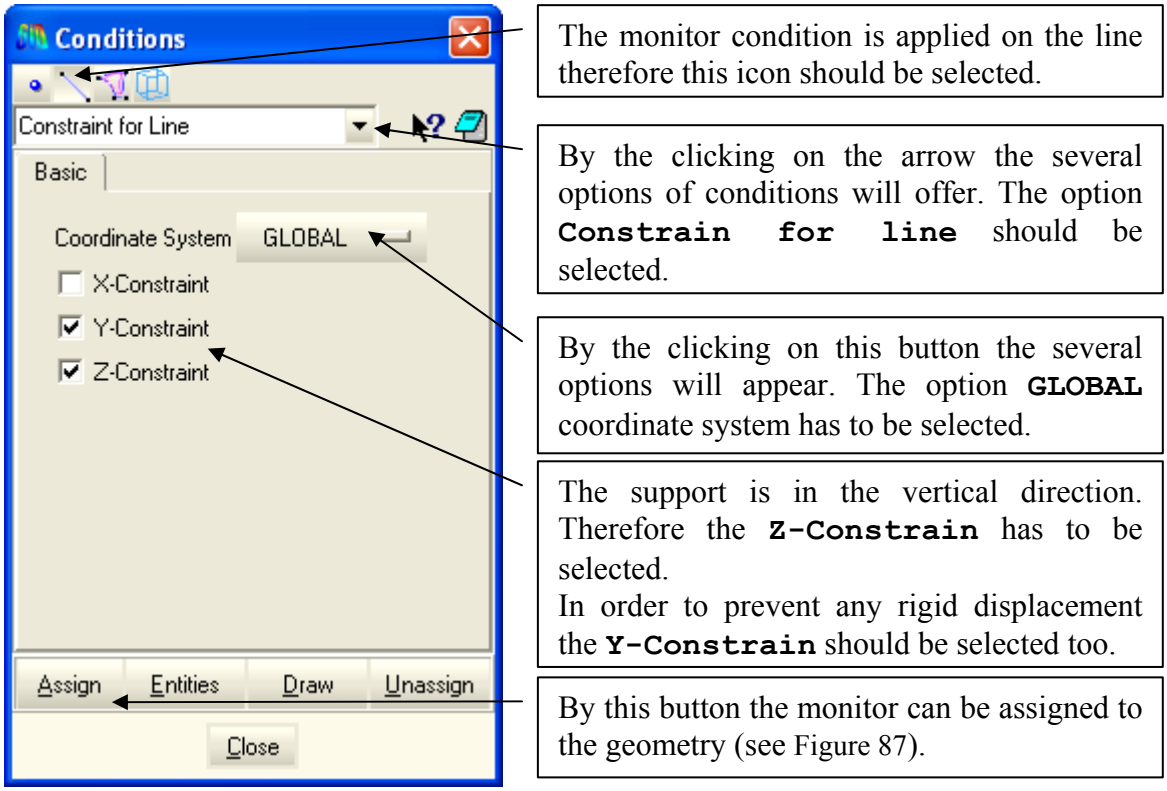

Conditions
Constraint for Line
Basic
Coordinate System
GLOBAL
X-Constraint
Y-Constraint
Z-Constraint
Assign
Entities
Draw
Unassign
Close
The monitor condition is applied on the line therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will offer. The option Constrain for line should be selected.
By the clicking on this button the several options will appear. The option GLOBAL coordinate system has to be selected.
The support is in the vertical direction. Therefore the Z-Constrain has to be selected.
In order to prevent any rigid displacement the Y-Constrain should be selected too.
By this button the monitor can be assigned to the geometry (see Figure 87).

Figure 86: The support condition definition

Parameter input:
Constraint for Line
Coordinate System: GLOBAL
Y-Constraint
Z-Constraint

By clicking on the icon the created condition can be drawn. After clicking on that icon the support condition will be displayed on the line (see Figure 88).

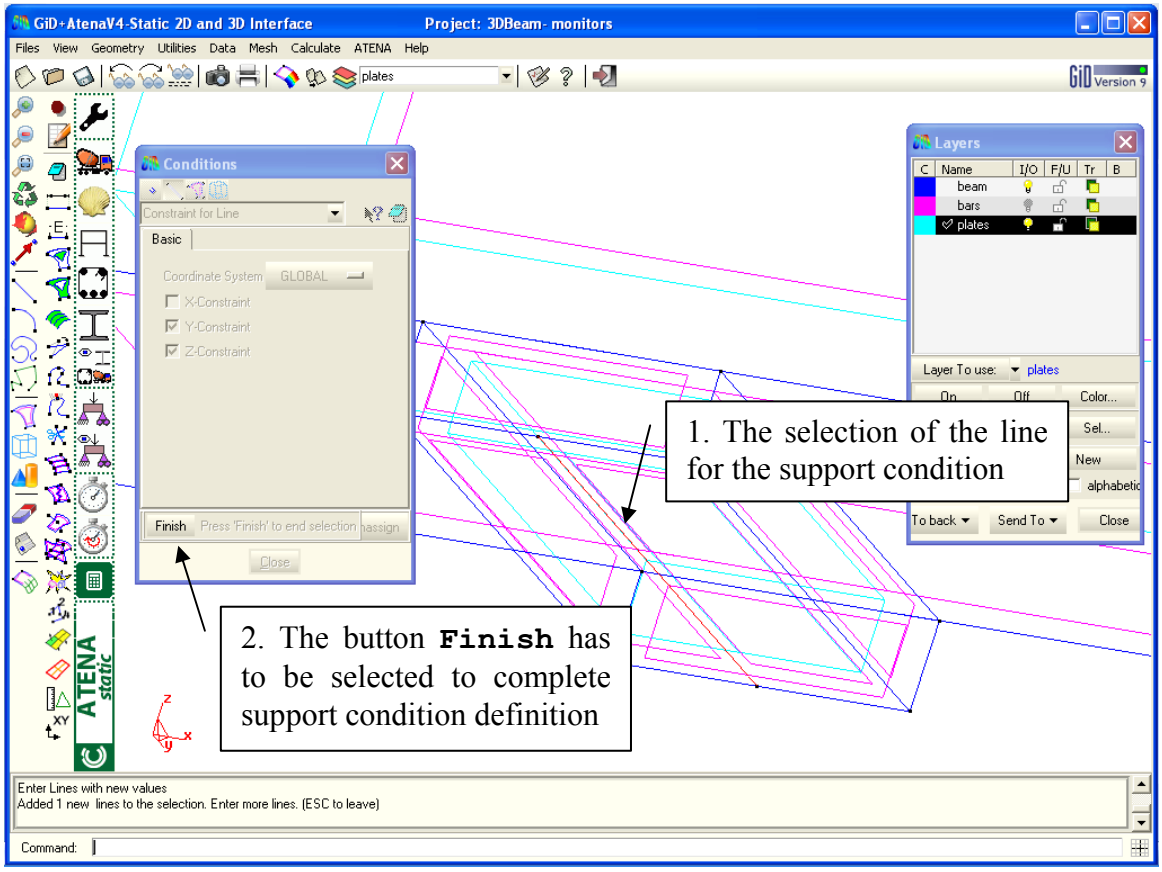


Figure 87: The selection of the support line

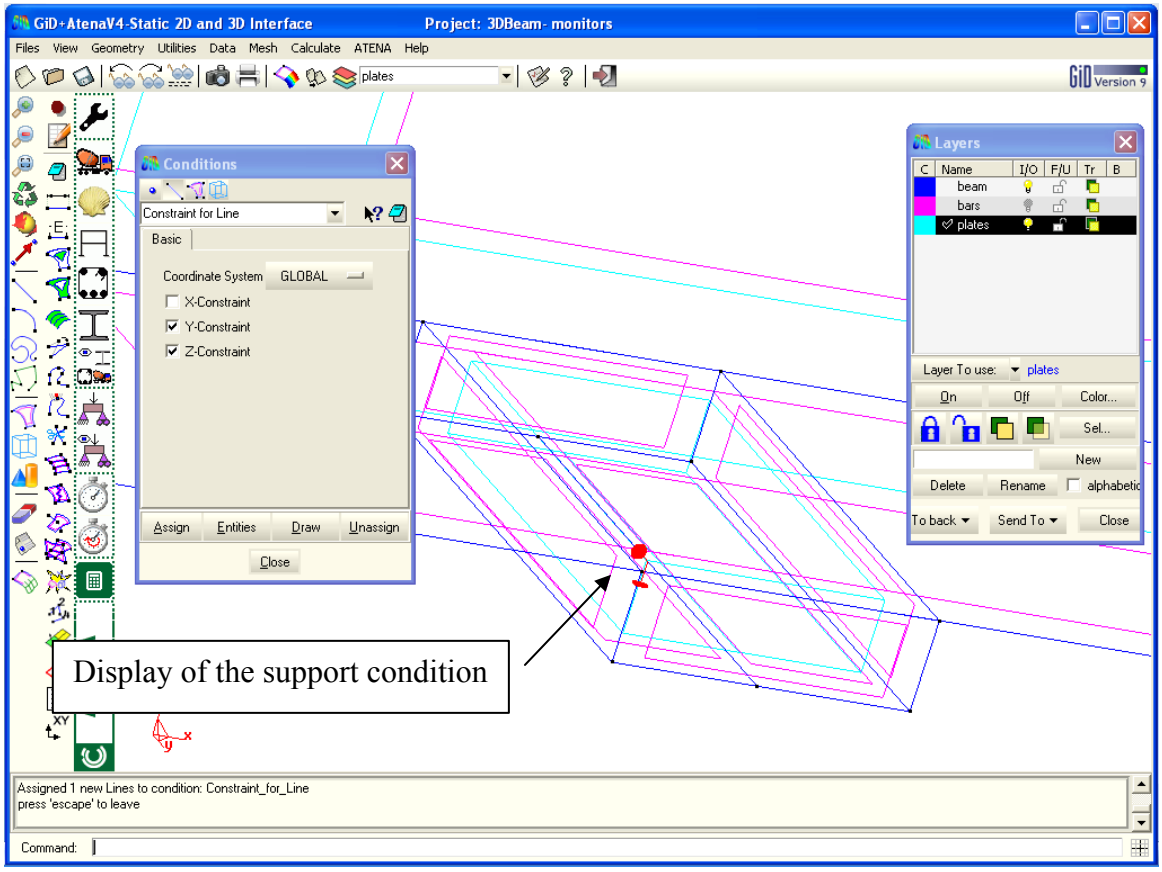


Figure 88: The support condition

3.4.2 Displacement

On the top plate the predefined displacement should be defined. This displacement will be located in the middle of the loading plate (top plate) and the displacement should be defined -0.0001m in the z direction.

This load should be applied on the point. This point is not created yet. Therefore first the geometry of the top plate has to be modified.

The point should lie in the centre of the top surface. This point has to be part of the top plate geometry. It cannot be simply created on the surface. Therefore the top surface will be divided into two surfaces and then the line which separates these surfaces will be also divided into two parts. Then the middle point will represent a monitor point and the monitor condition can be applied.

The steel plates are placed in the plates layer. Therefore the plate layer should be activated and displayed. The beam and bars layers can be hidden. It is also recommended to zoom in the top plate (see Figure 89).

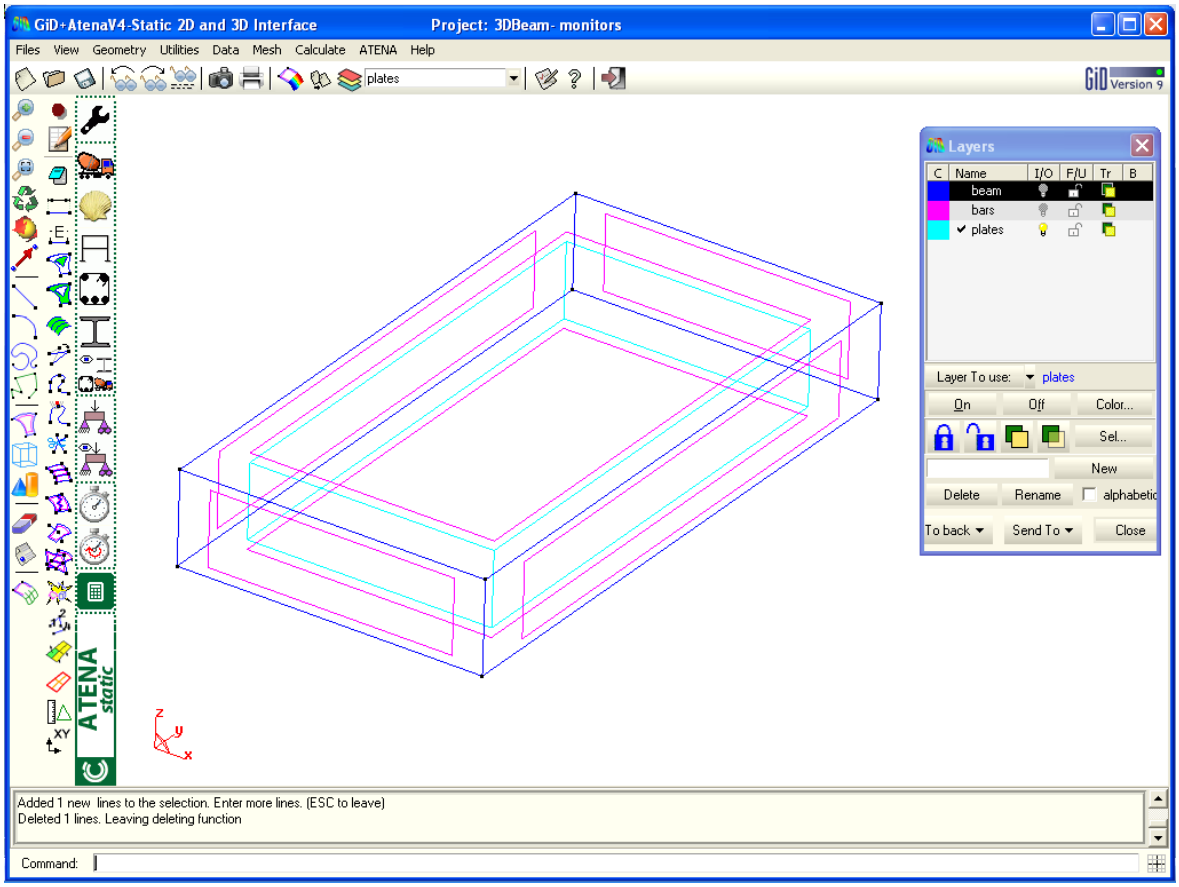


Figure 89: The activated plates layer and zoomed top plate

The to surface will be devided with executing of the command from main menu **Geometry | Edit | Divide| Surfaces | Num Division** or by selecting of the Divide surface icon (see Figure 90).

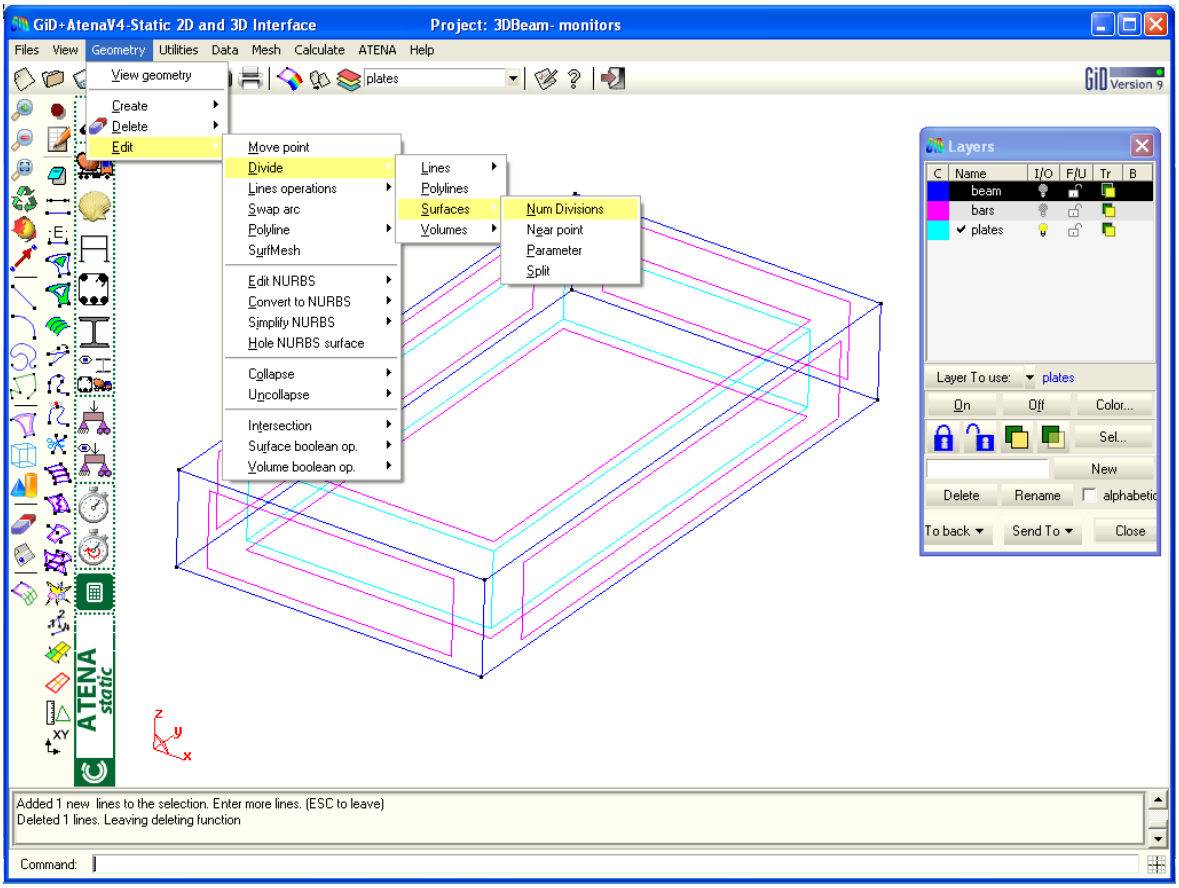


Figure 90: The executing of the division command

After the executing the divide command the cursor will change into this shape and the surface required for dividing should be selected. Once the surface is selected the dialog window will appear on the screen (see Figure 91).

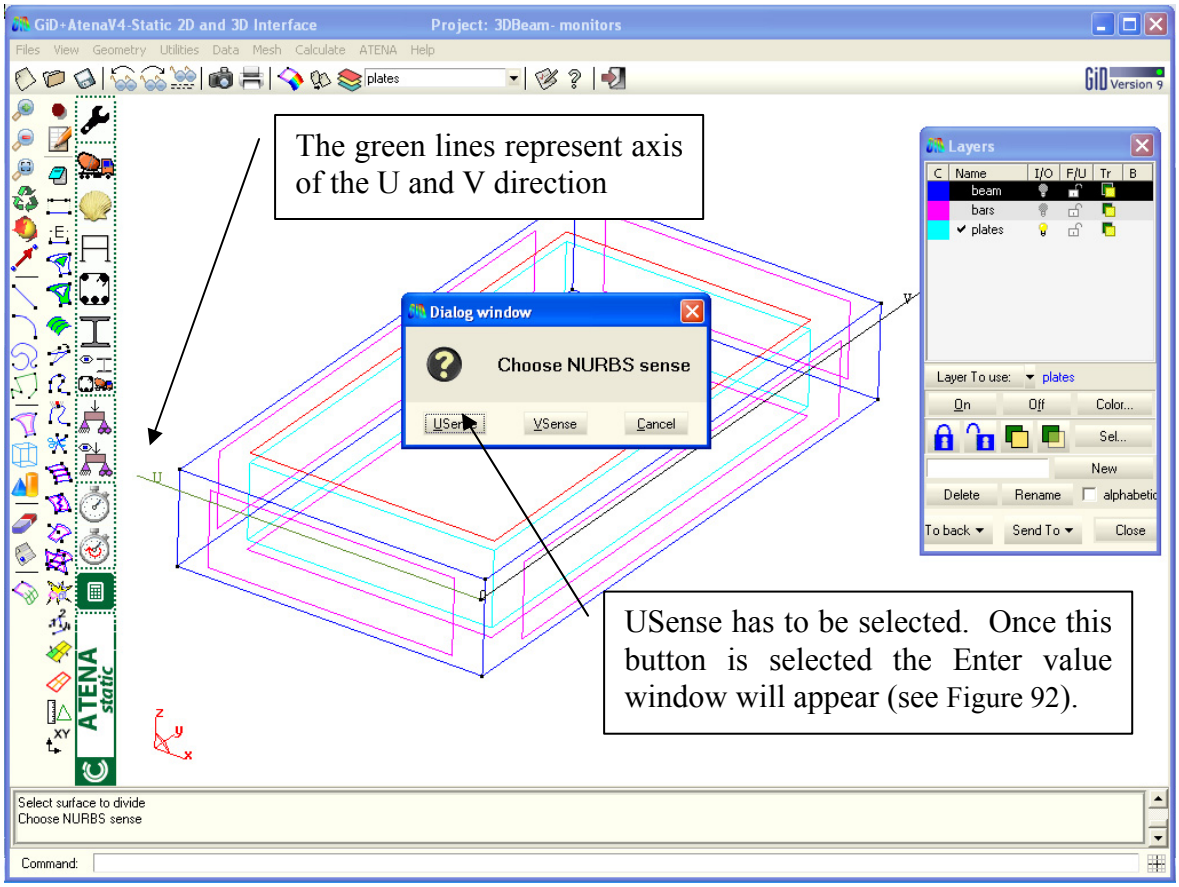


Figure 91: The dividing of the surface

The dialog window asks in which direction the surface should be divided. There are U and V direction and in the graphical area it is possible to see green axis representing U and V direction. In this case USense should be chosen. Once the USense button is chosen the program asks for the number of the divisions. Top surface should be divided into two parts (see Figure 92).

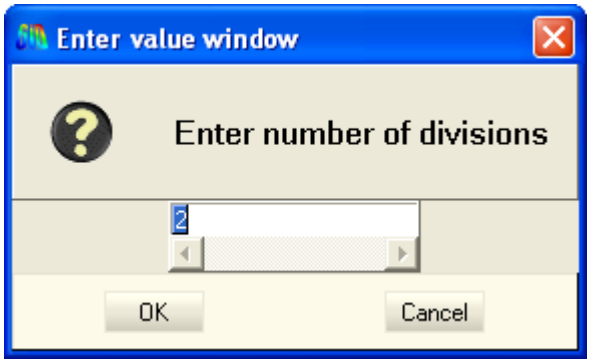


Figure 92: The enter value window

Parameter input:
Enter number of divisions: 2

The button **OK** should be pressed in the enter value window. After that the surfaces is divided (see Figure 93).

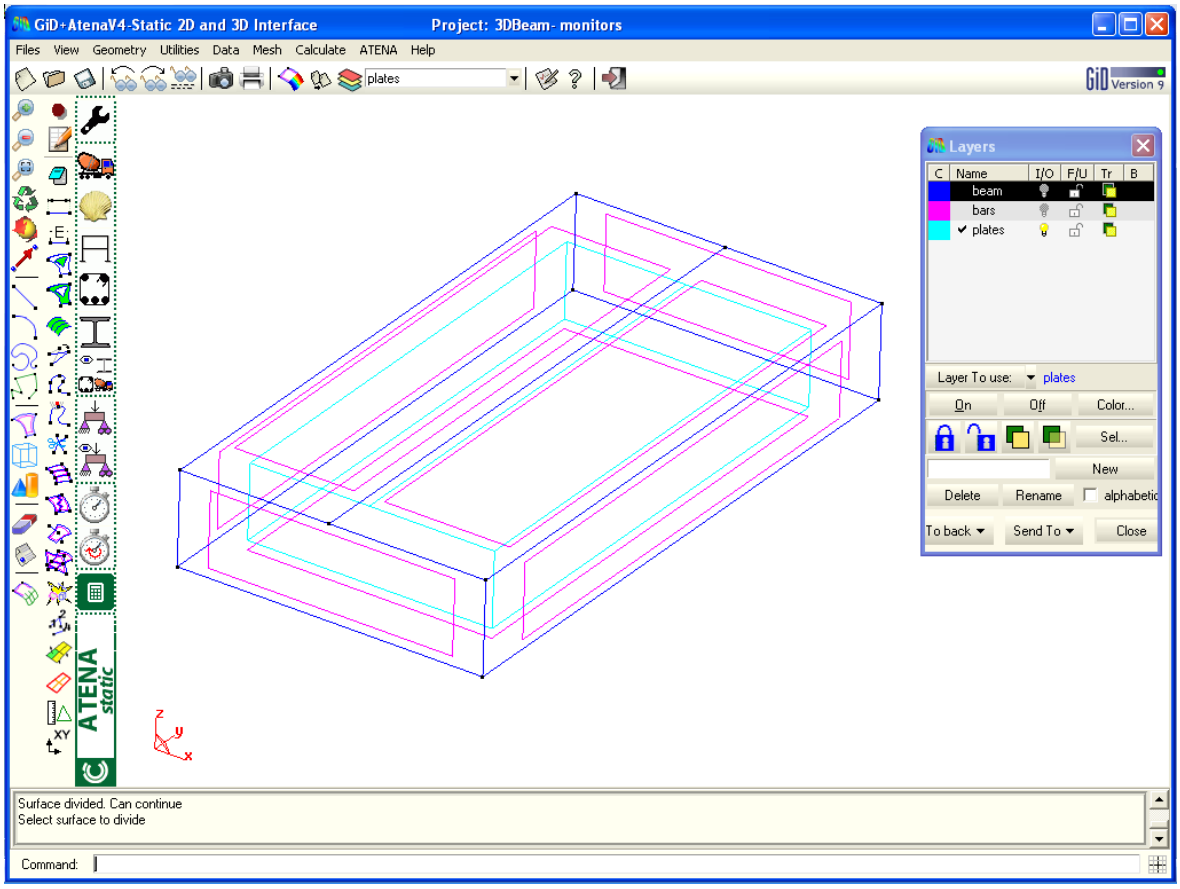


Figure 93: The divided top surface

Now the middle line can be divided into two parts. It can be done by executing command **`Geometry | Edit | Divide| Lines | Num Division`** or by the icon . After the executing of this command the enter value window will appear. There should be written number of divisions. The line should be divided in two divisions (see Figure 95).

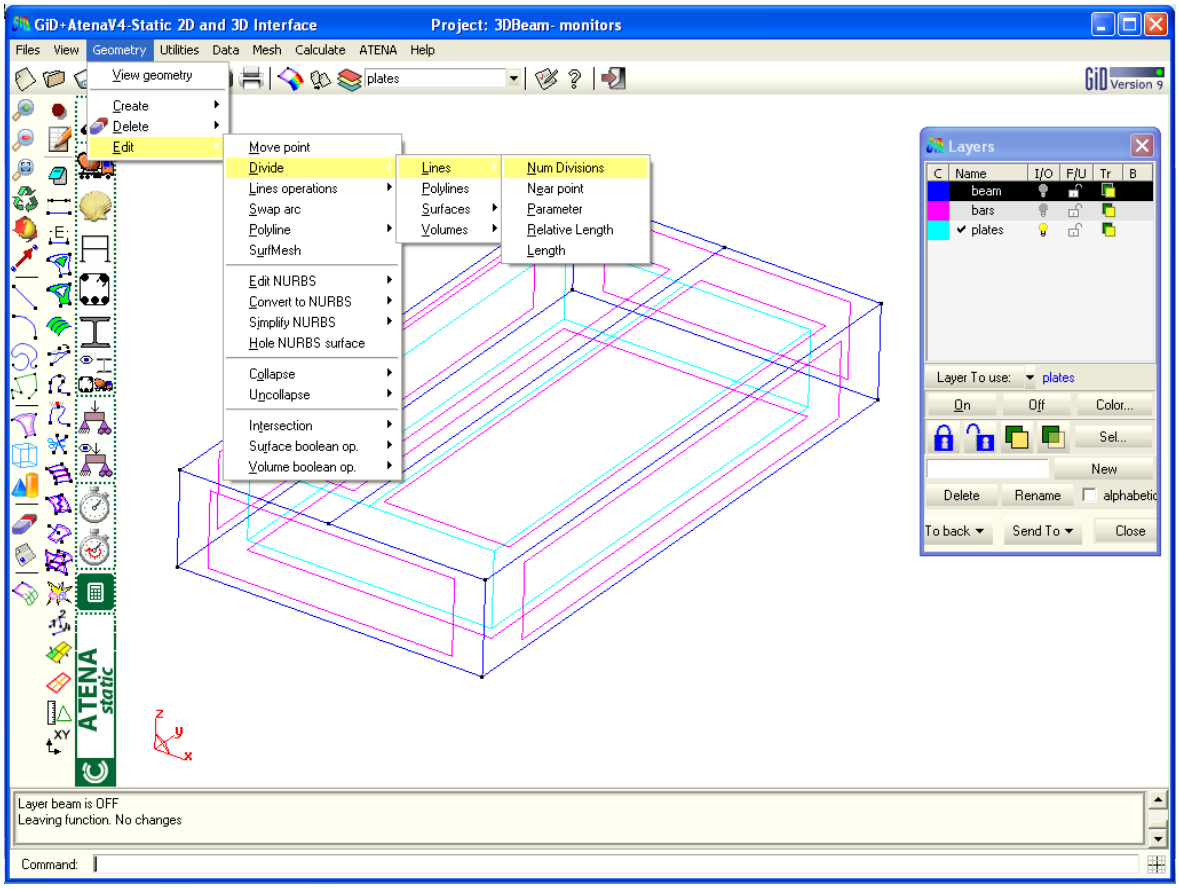


Figure 94: The dividing of the line

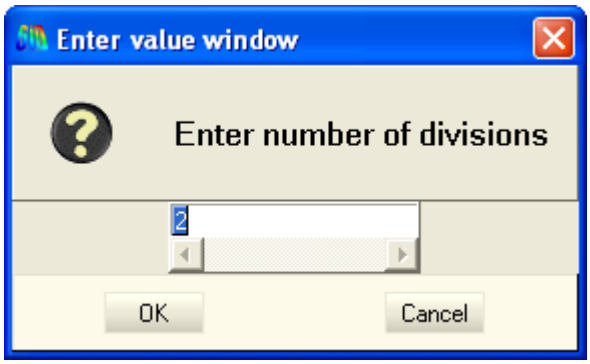


Figure 95: The enter value window

Parameter input:
Enter number of divisions: 2

For the selection of the line which should be divided the button **`OK`** has to be pressed. After the selection (see Figure 96) of the line the ESC key has to be pressed to complete this command (see Figure 97).

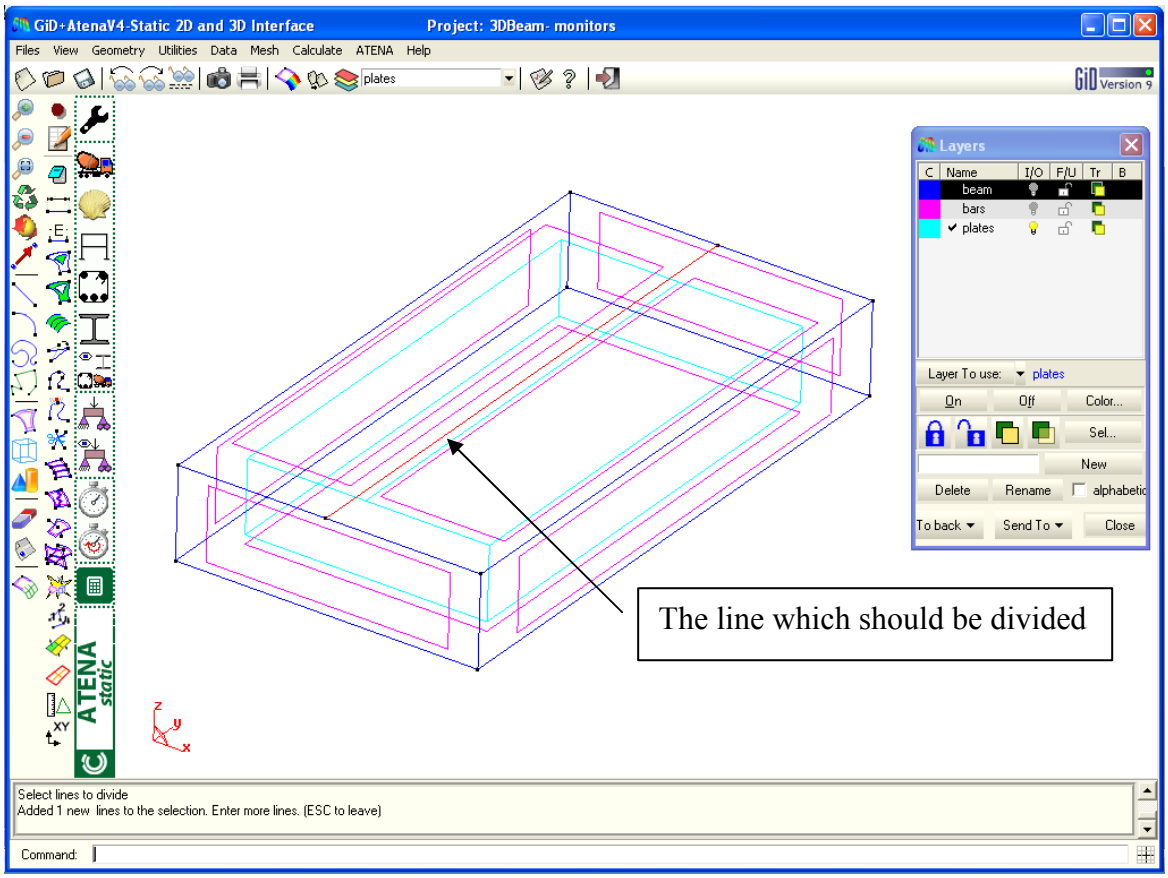


Figure 96: The selection of the line

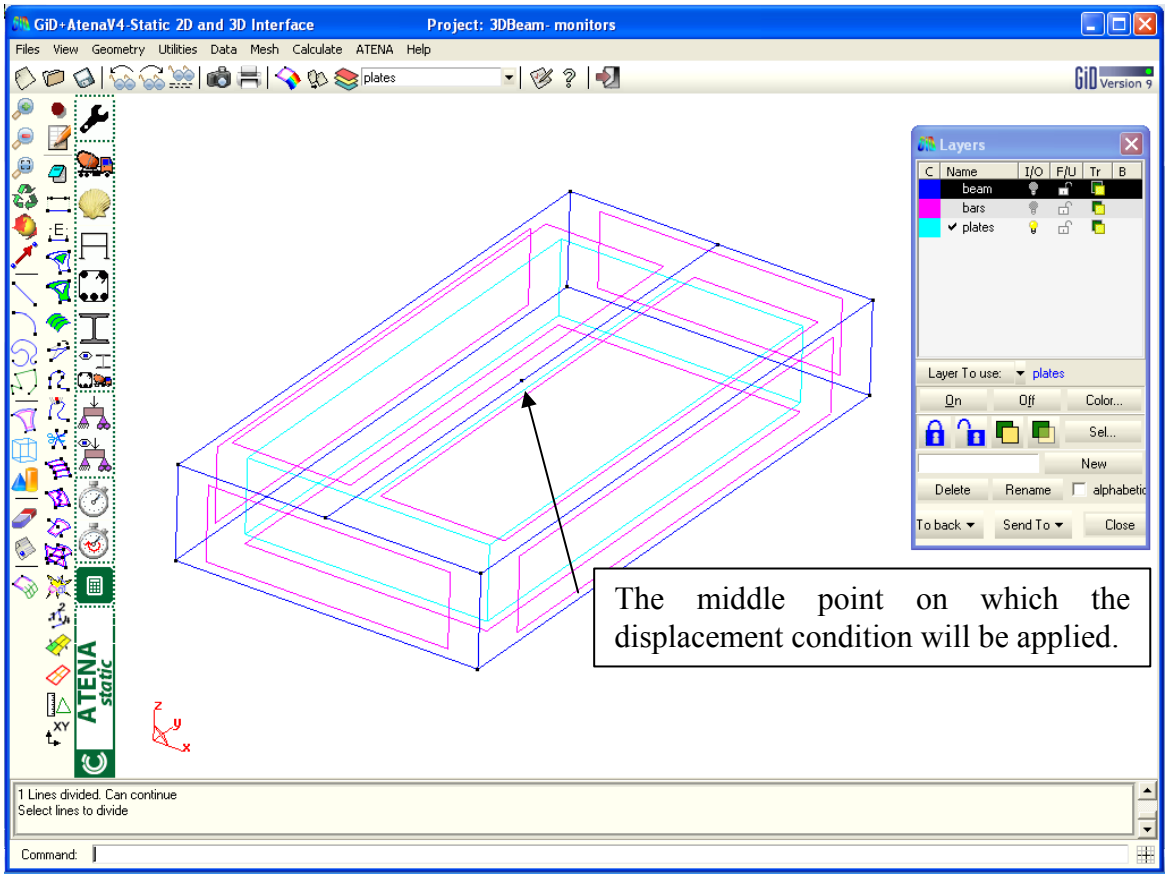


Figure 97: The divided line

The geometry for the displacement condition is already created. Any boundary conditions are assigned to the geometry and then depends on the geometry in which layer is located. Therefore it is not necessary to control which layer is activated.

Conditions command can be executed by the icon or by **`Data | Conditions`** in the main menu. The displacement condition definition is depicted in the Figure 98.

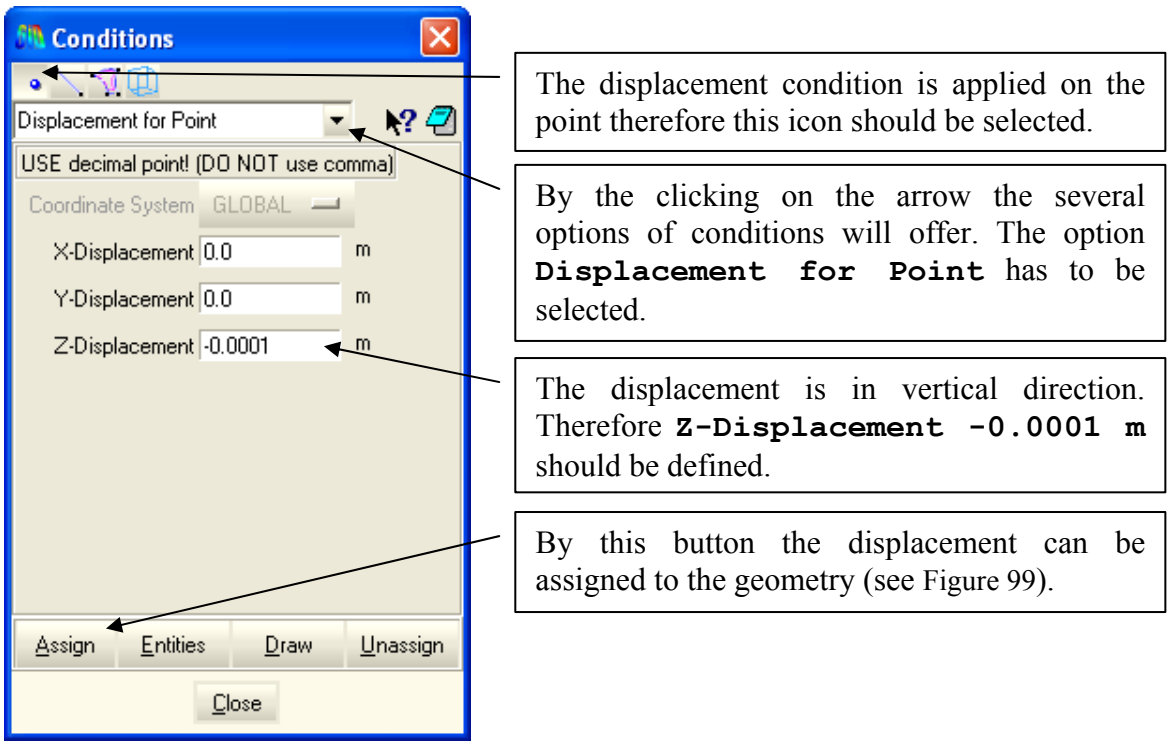

Conditions
Displacement for Point
USE decimal point! (DO NOT use comma)
Coordinate System GLOBAL
X-Displacement 0.0 m
Y-Displacement 0.0 m
Z-Displacement -0.0001 m
Assign
Entities
Draw
Unassign
Close
The displacement condition is applied on the point therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will offer. The option Displacement for Point has to be selected.
The displacement is in vertical direction. Therefore Z-Displacement -0.0001 m should be defined.
By this button the displacement can be assigned to the geometry (see Figure 99).

Figure 98: The displacement condition definition

Parameter input:
Displacement for Point
Z-Displacement: -0.0001 m

By clicking on the icon the created condition can be drawn. After clicking on that icon the displacement condition will be displayed on the point (see Figure 100).

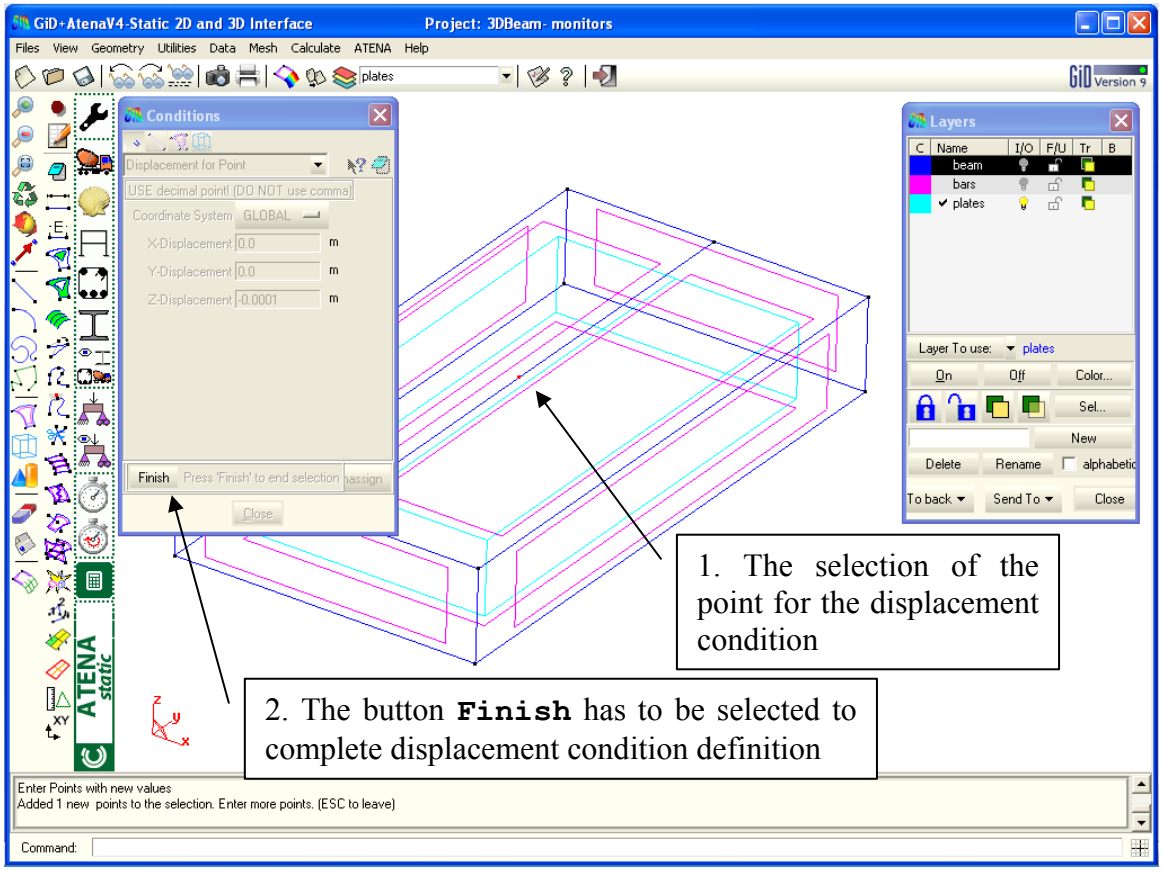

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- monitors
Files View Geometry Utilities Data Mesh Calculate ATENA Help
plates
Conditions
Displacement for Point
USE decimal point! (DO NOT use comma)
Coordinate System GLOBAL
X-Displacement 0.0 m
Y-Displacement 0.0 m
Z-Displacement -0.0001 m
Finish Press 'Finish' to end selection
Close
Layers
C Name I/O F/U Tr B
beam
bars
plates
Layer To use: plates
On Off Color...
Sel...
New
Delete Rename alphabetic
To back Send To Close
1. The selection of the point for the displacement condition
2. The button Finish has to be selected to complete displacement condition definition
Enter Points with new values
Added 1 new points to the selection. Enter more points. (ESC to leave)
Command:

Figure 99: The selection of the point for the displacement condition

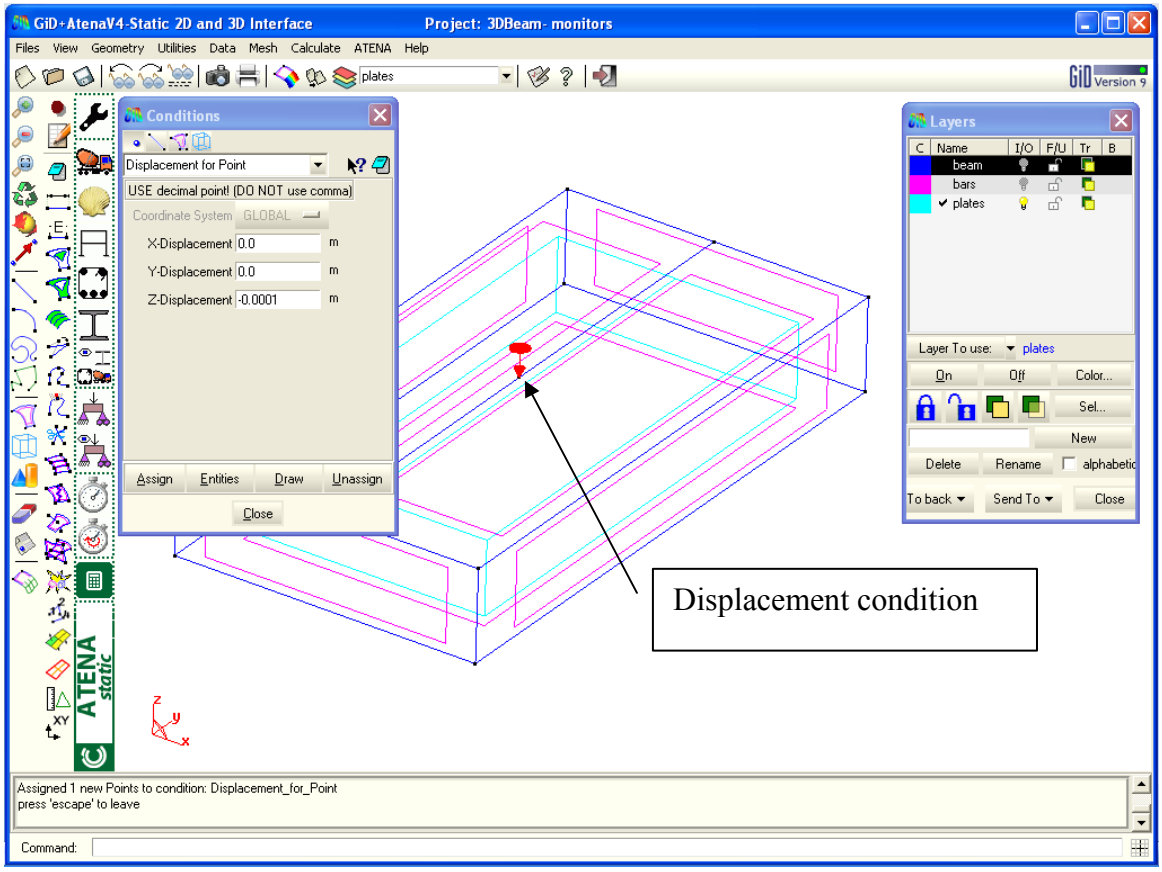


Figure 100: The displacement condition

3.4.3 Symmetry condition

The beam of this example is symmetrical. Therefore the only half of the beam is analysed and it is necessary to enforce the axis of the symmetry along right side of the beam. This means that the horizontal x-displacements along this side should be equal to zero. It can be done by definition of the boundary condition on the surface (see Figure 101).

Conditions command can be executed by the icon or by the **`Data | Conditions`** in the main menu. The symmetry condition definition is depicted in the Figure 102.

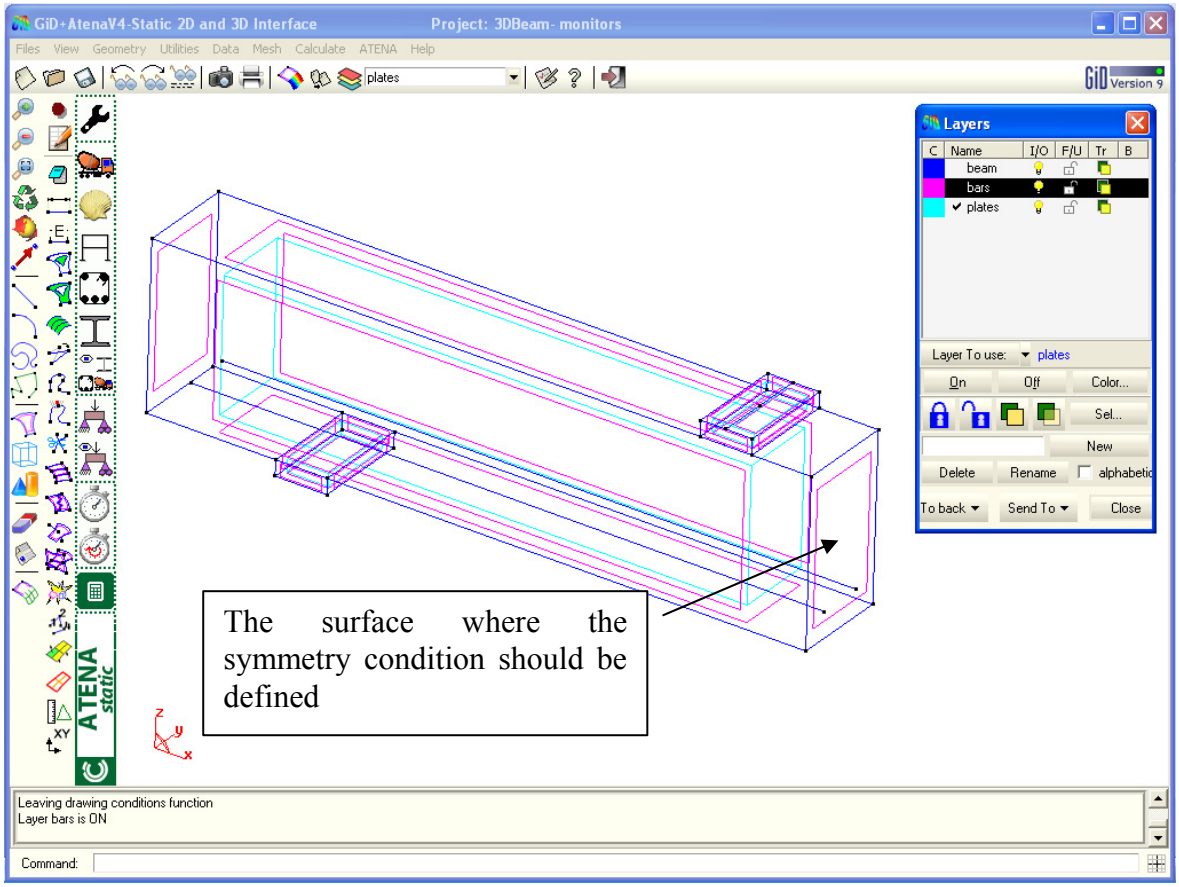


Figure 101: The surface for the symmetry condition

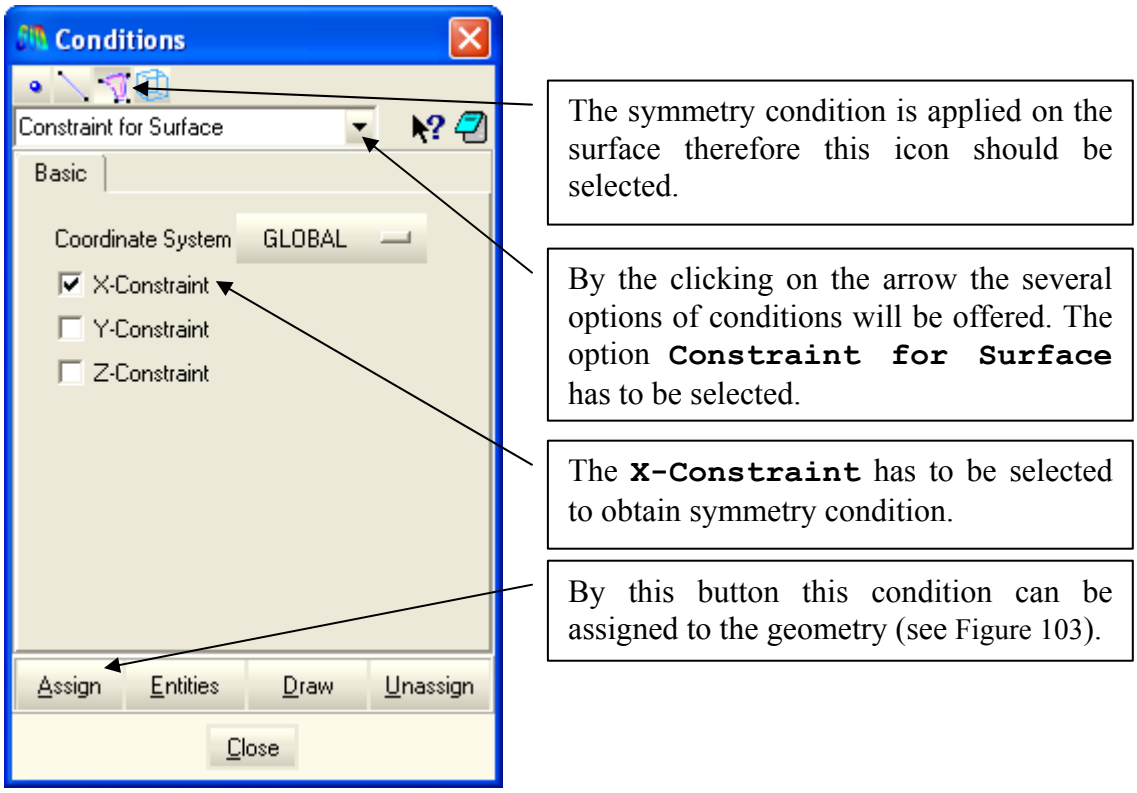

Conditions
Constraint for Surface
Basic
Coordinate System
GLOBAL
X-Constraint
Y-Constraint
Z-Constraint
Assign
Entities
Draw
Unassign
Close
The symmetry condition is applied on the surface therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will be offered. The option Constraint for Surface has to be selected.
The X-Constraint has to be selected to obtain symmetry condition.
By this button this condition can be assigned to the geometry (see Figure 103).

Figure 102: The symmetry condition definition

Parameter input:
Constraint for Surface
Coordinate System: GLOBAL
X-Constraint

By clicking on the icon the created condition can be drawn. After clicking on that icon the symmetry condition will be displayed on the surface (see Figure 104).

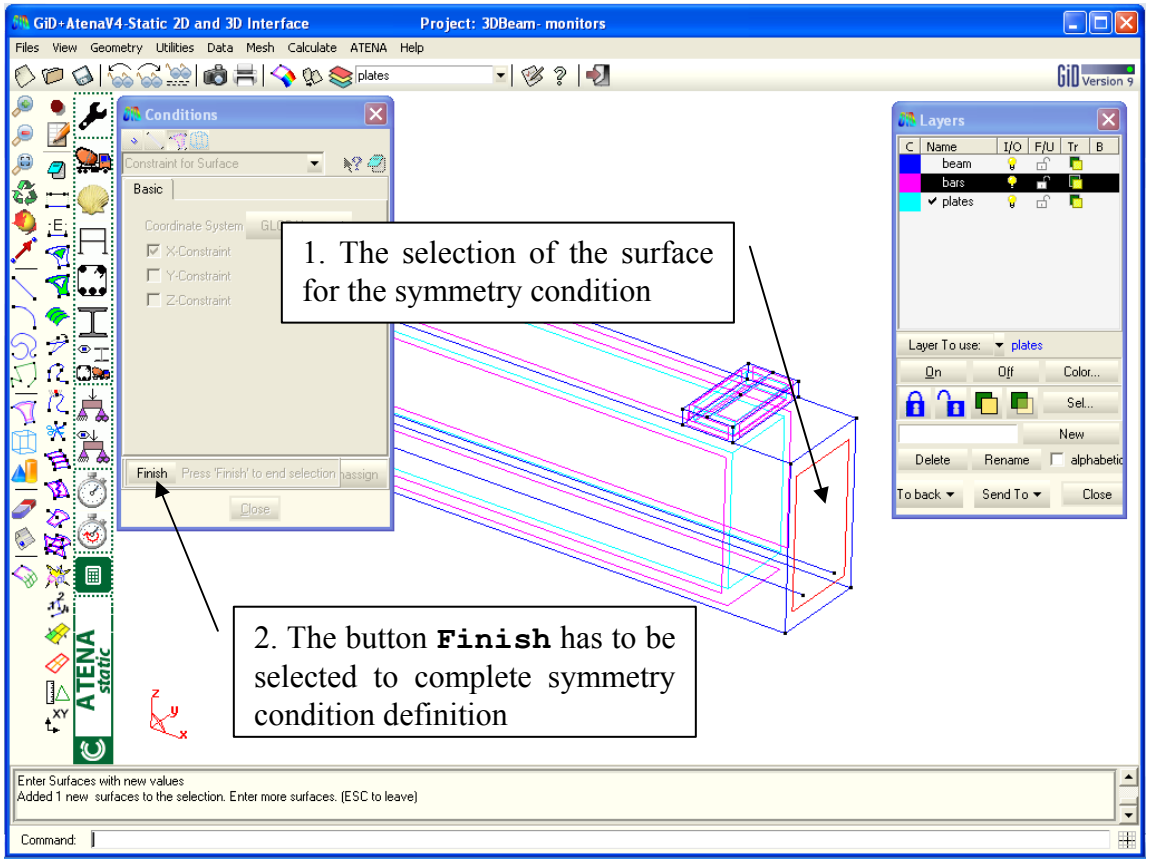


Figure 103: The selection of the surface for the symmetry condition

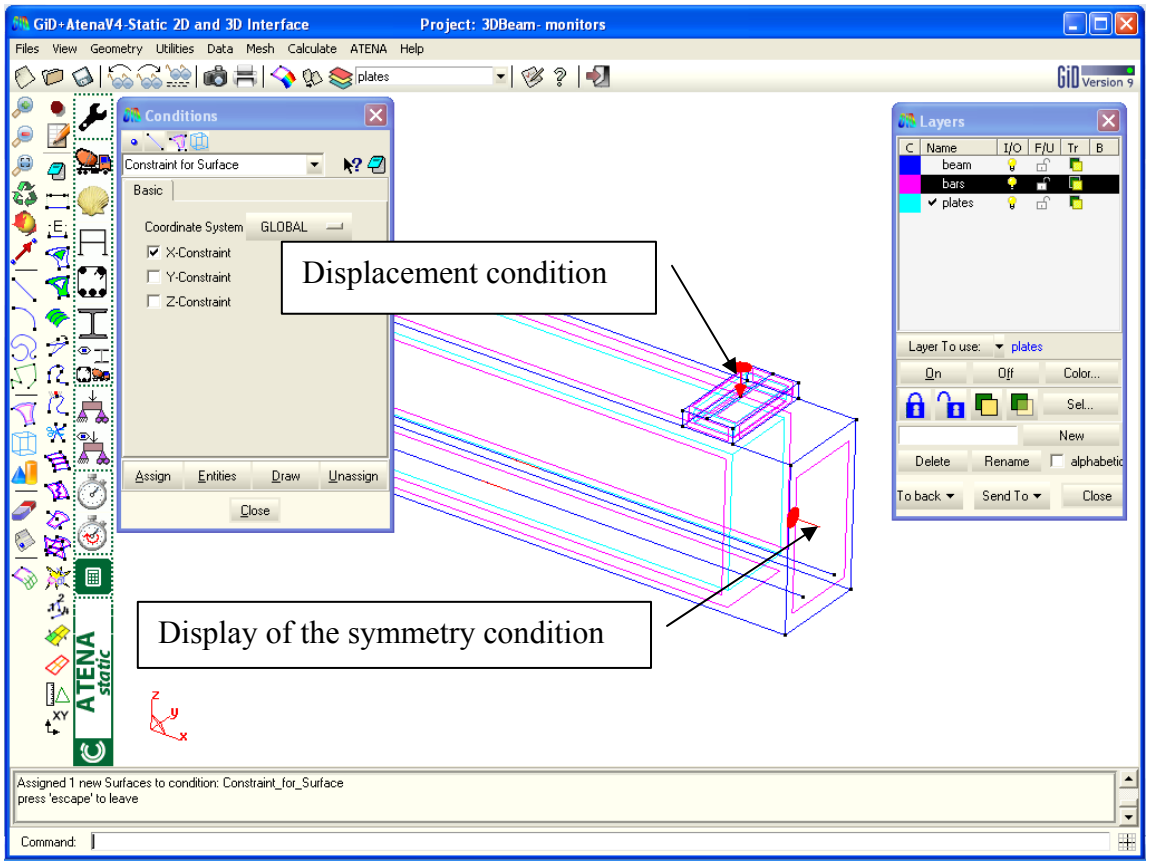


Figure 104: The symmetry condition

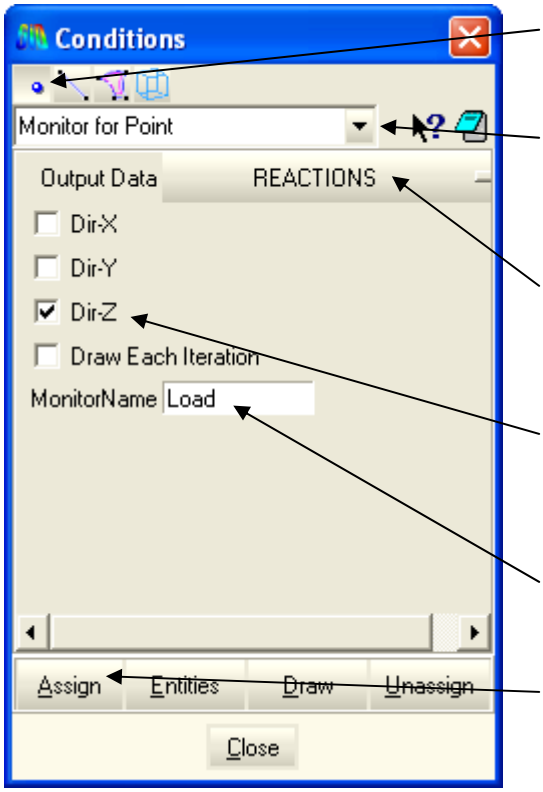
3.4.4 Monitors

Monitors provide important information about state of the structure because monitor can monitor different states of the particular geometrical entity. On that entity the monitor condition has to be applied.

In this example the points are used for application of the monitor condition. In this case two monitors will be defined. Geometry of one of these points is already created but the geometry of the second monitor has to be created.. Therefore the geometry has to be modified. One monitor will be monitoring loads on the top plate and second one will monitor deflections on the beam near its bottom surface.

3.4.4.1 First monitor

First monitor should be located on the top plate. It will be applied on the point where the displacement condition is also defined. The definition of monitor condition starts by the icon or by executing command **`Data | Conditions`** in the main menu. The monitor condition definition is depicted in the Figure 105.



The monitor condition is applied on the point therefore this icon should be selected.

By the clicking on the arrow the several options will offer. The option **`Monitor for point`** has to be selected.

By the clicking on this button the several options will appear. The option **`REACTIONS`** has to be selected

The monitor point will monitor reactions in the **`Z direction`** therefore this checkbox has to be selected.

The name of the top plate monitor will be **`Load`**.

By this button the monitor can be assigned to the geometry (see Figure 106).

Figure 105: The first monitor condition definition

Parameter input:
Monitor for point
Output Data: REACTIONS
Dir-Z
Monitor Name: Load

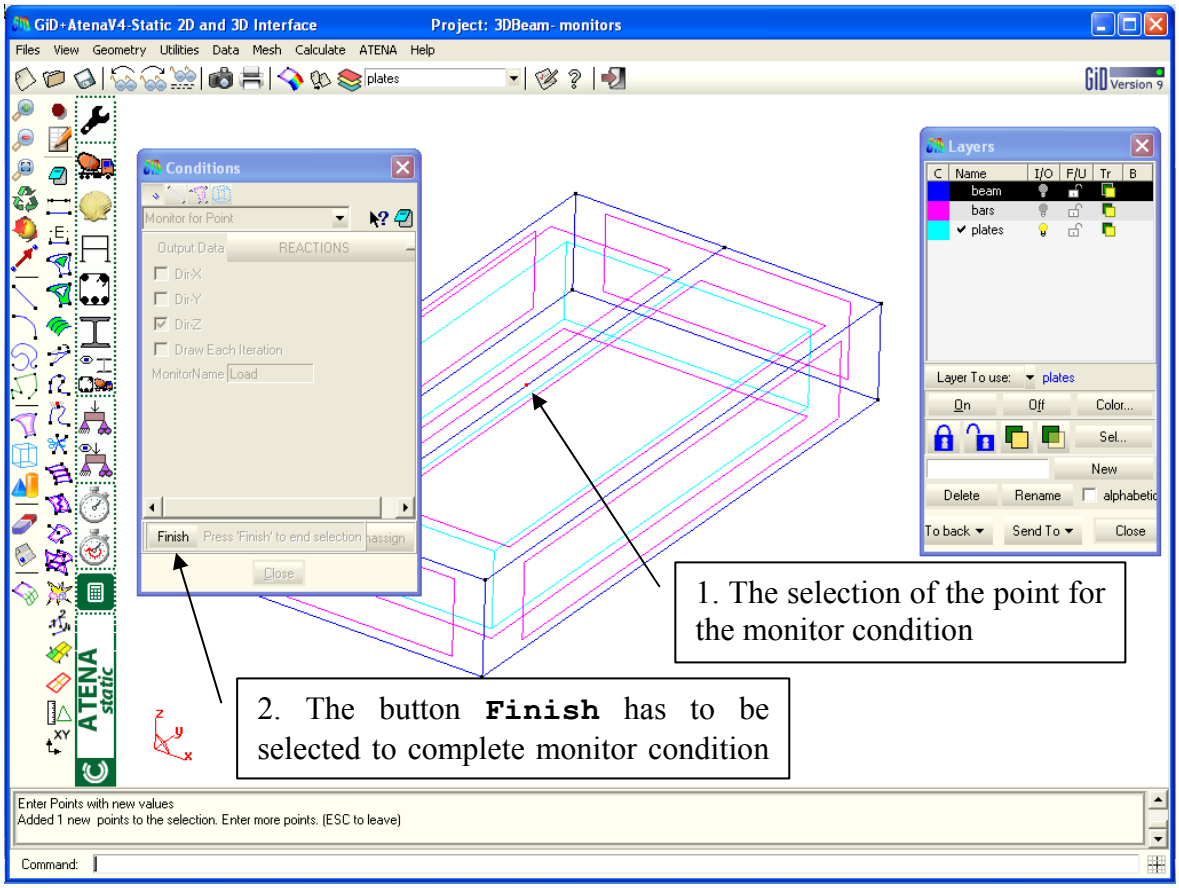


Figure 106: The selection of the first monitoring point

By clicking on the icon the created condition can be drawn. After clicking on that icon the monitor condition will be displayed on the point (see Figure 107).

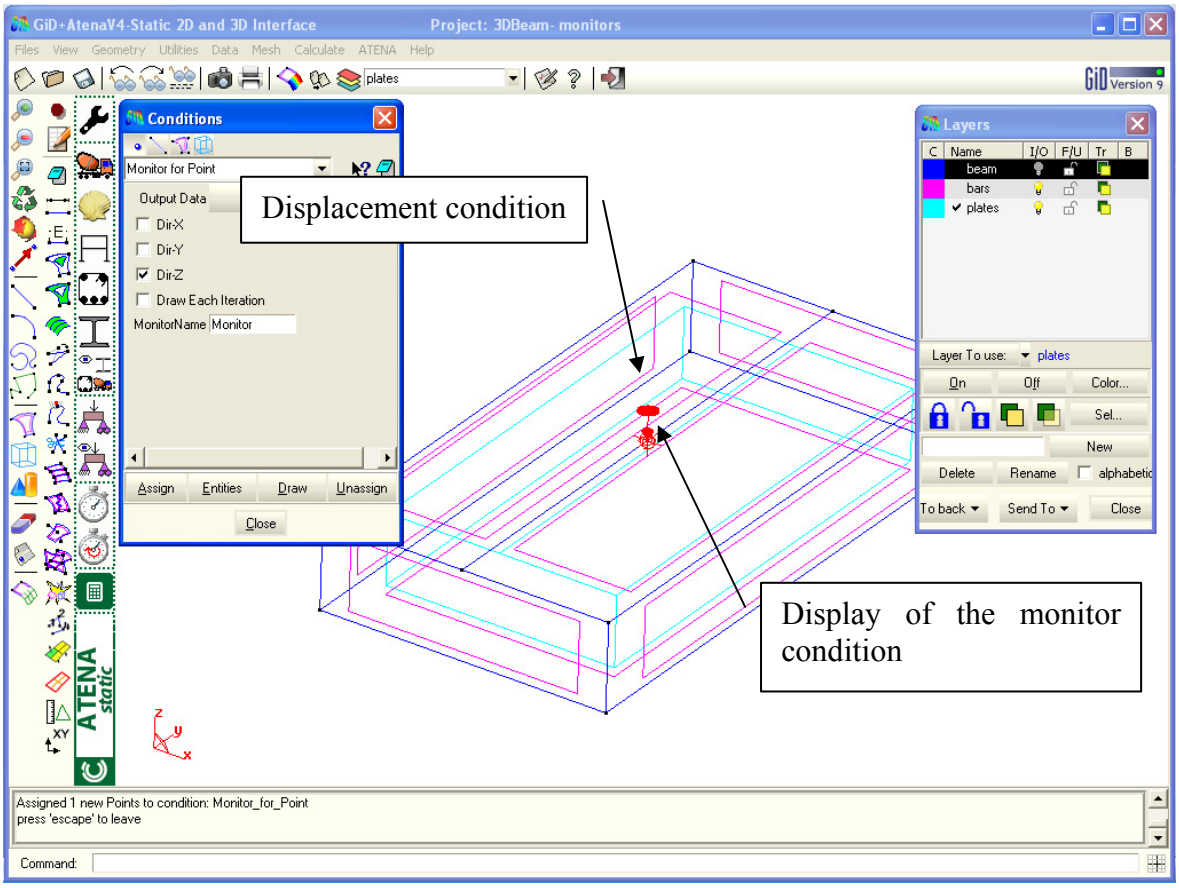


Figure 107: The first monitor condition

3.4.4.2 Second monitor

The second monitor point should be located at the middle of the beam near its bottom surface, where the largest vertical displacement can be expected. First the geometry of the point has to be created. It will be done by the division of the down-right edge of the beam.

Before starting it is better to display only the beam layer and zoom in the down-right edge of the beam. The procession of the division will be same like in previous section.

The division command can be executed by the icon or from the main menu by **`Geometry | Edit | Divide| Lines | Num Division`**(see Figure 108).

After the executing of this command the enter value window will appear. There should be written number of divisions. The line should be divided in two divisions (see Figure 109).

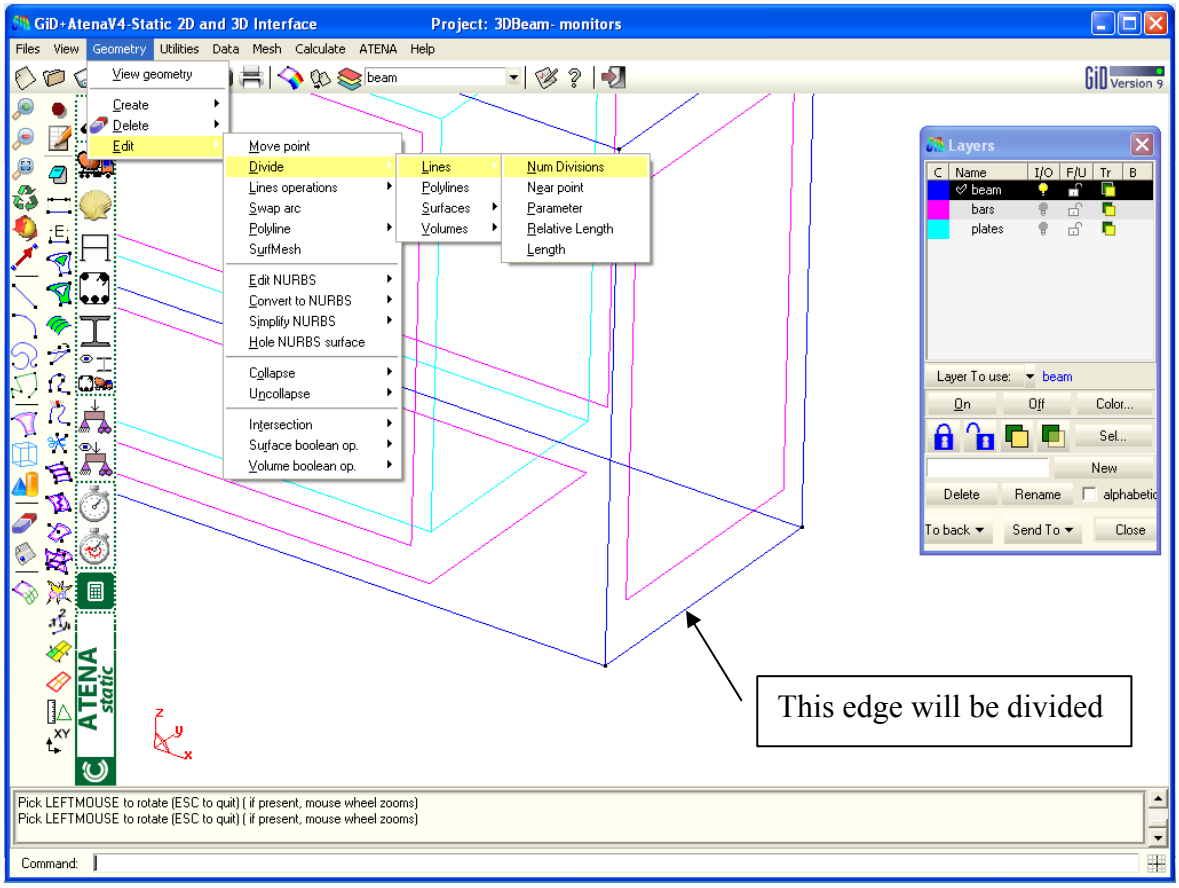


Figure 108: The executing of the division command

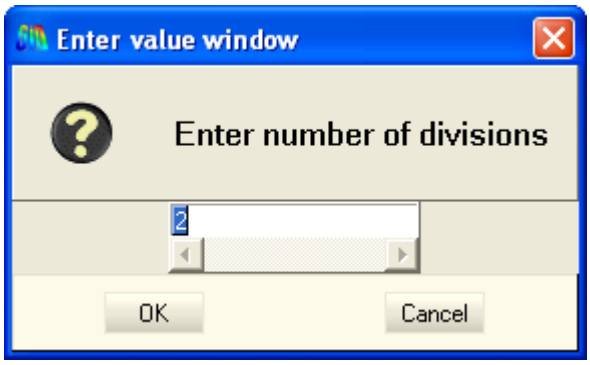


Figure 109: The enter value window

Parameter input:
Enter number of divisions: 2

For the selection of the line which should be divided the button **OK** has to be pressed. After the selection of the line the ESC key has to be pressed to complete this command (see Figure 110).

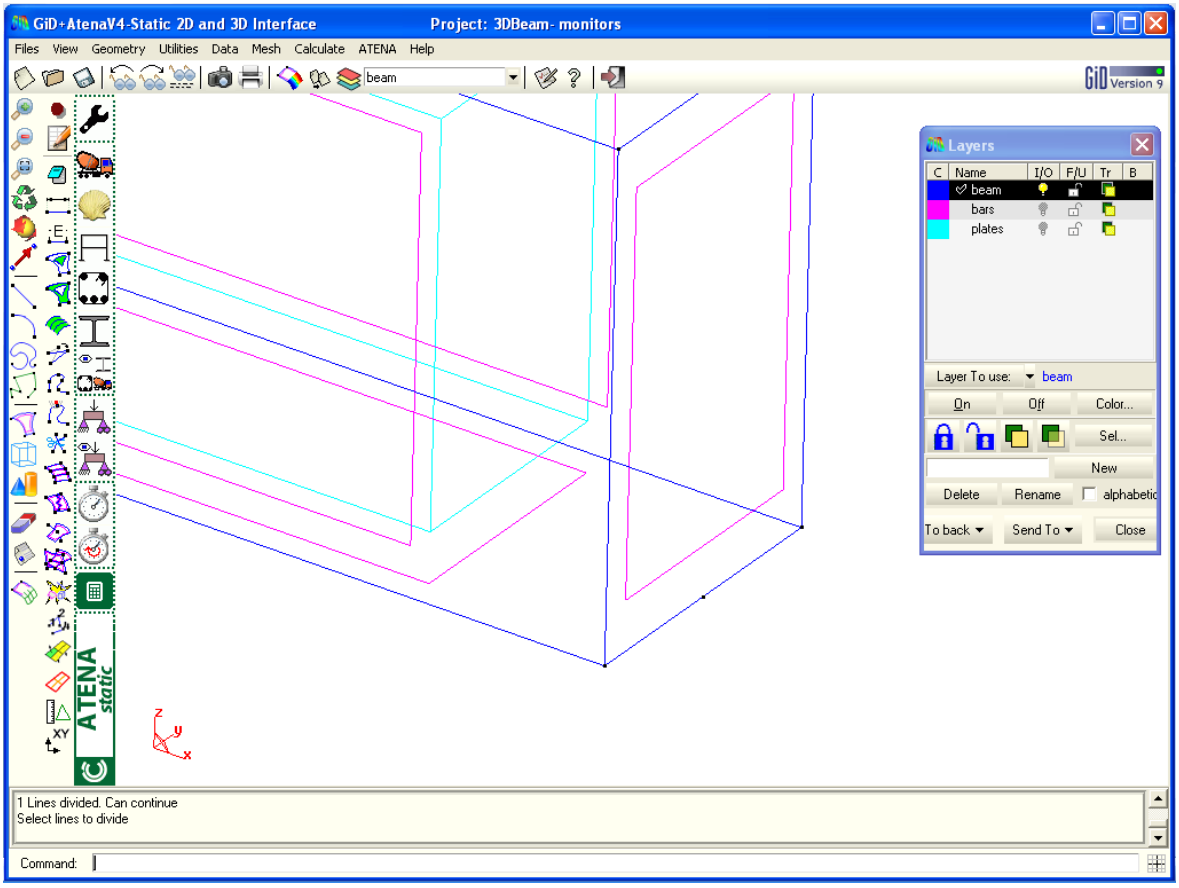


Figure 110: The created second monitor point

When the geometry for the monitoring point is created the monitor condition can be defined. Conditions command can be executed by the **Data | Conditions** in the main menu or by the icon . The second monitor condition definition is depicted in the Figure 111.

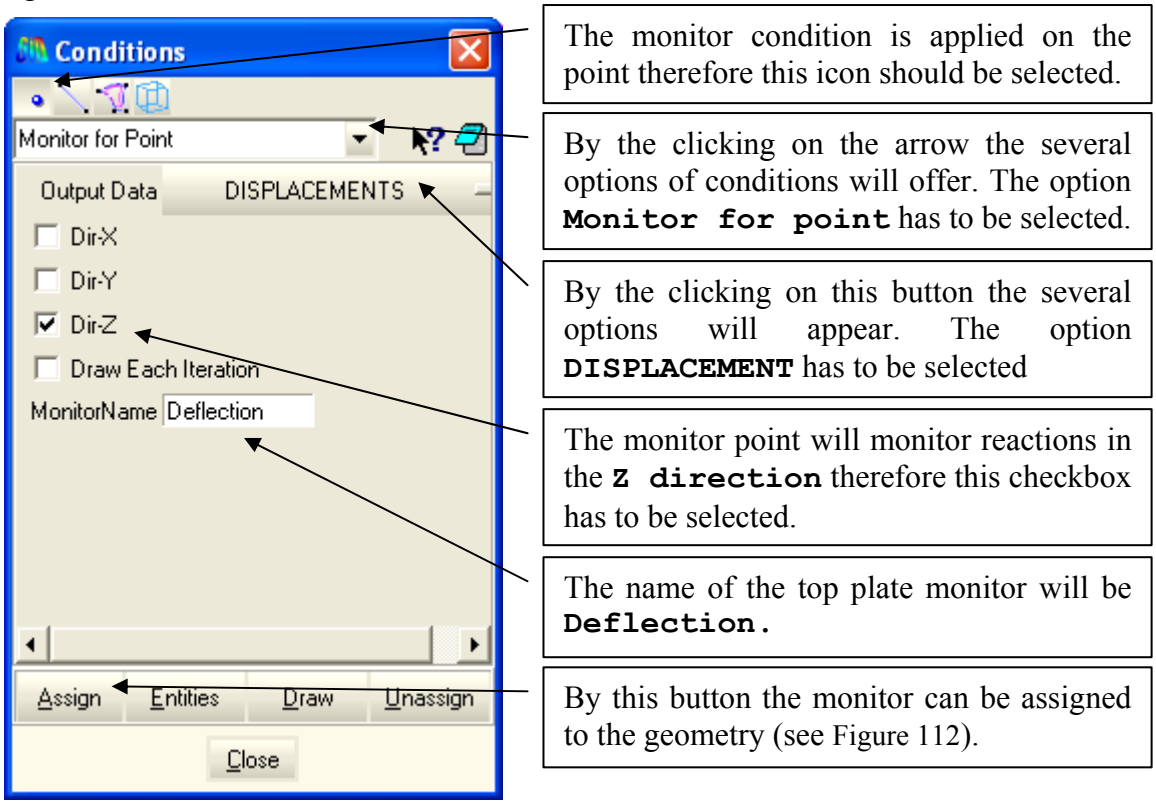

Conditions
Monitor for Point
Output Data
DISPLACEMENTS
Dir-X
Dir-Y
Dir-Z
Draw Each Iteration
MonitorName
Deflection
Assign
Entities
Draw
Unassign
Close
The monitor condition is applied on the point therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will offer. The option Monitor for point has to be selected.
By the clicking on this button the several options will appear. The option DISPLACEMENT has to be selected
The monitor point will monitor reactions in the Z direction therefore this checkbox has to be selected.
The name of the top plate monitor will be Deflection.
By this button the monitor can be assigned to the geometry (see Figure 112).

Figure 111: The second monitor condition definition

Parameter input:
Monitor for point
Output Data: DISPLACEMENT
Dir-Z
Monitor Name: Deflection

By clicking on the icon the created condition can be drawn. After clicking on that icon the monitor condition will be displayed on the point (see Figure 113).

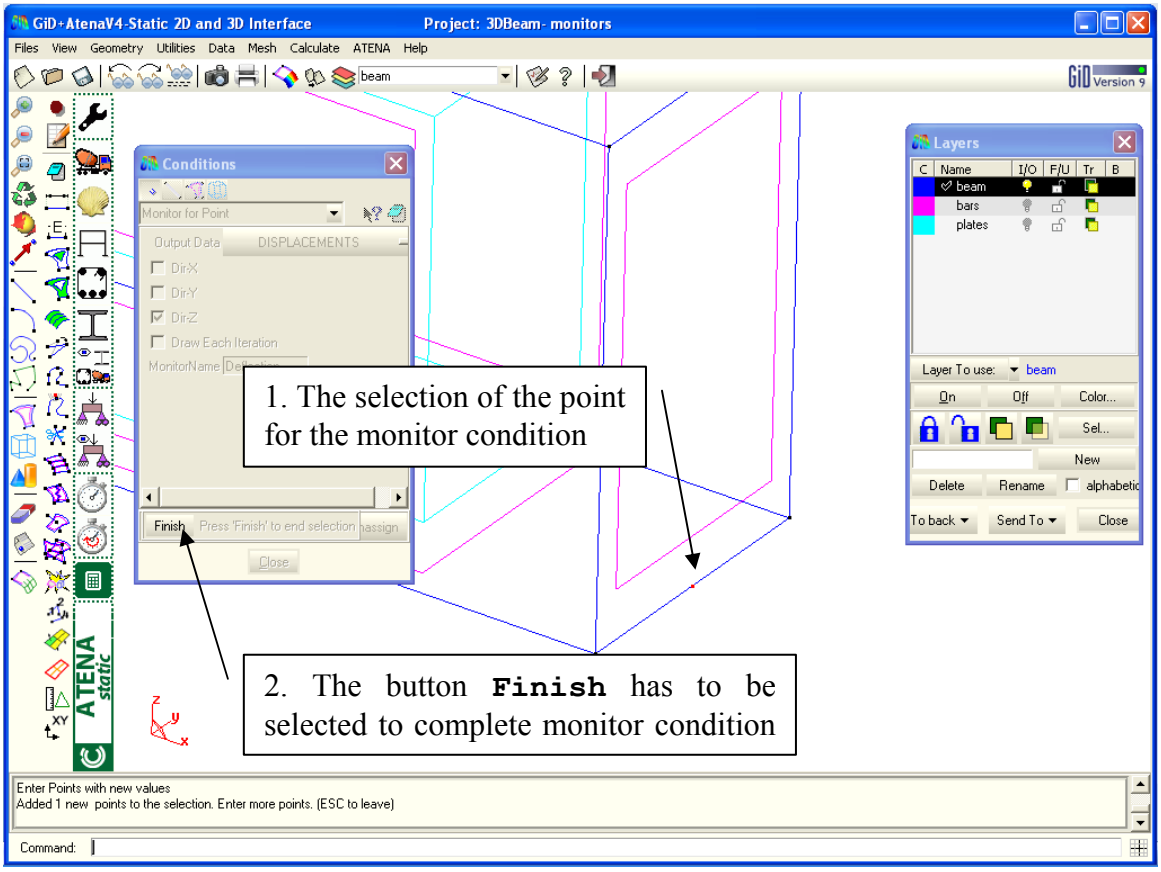


Figure 112: The selection of the second monitoring point

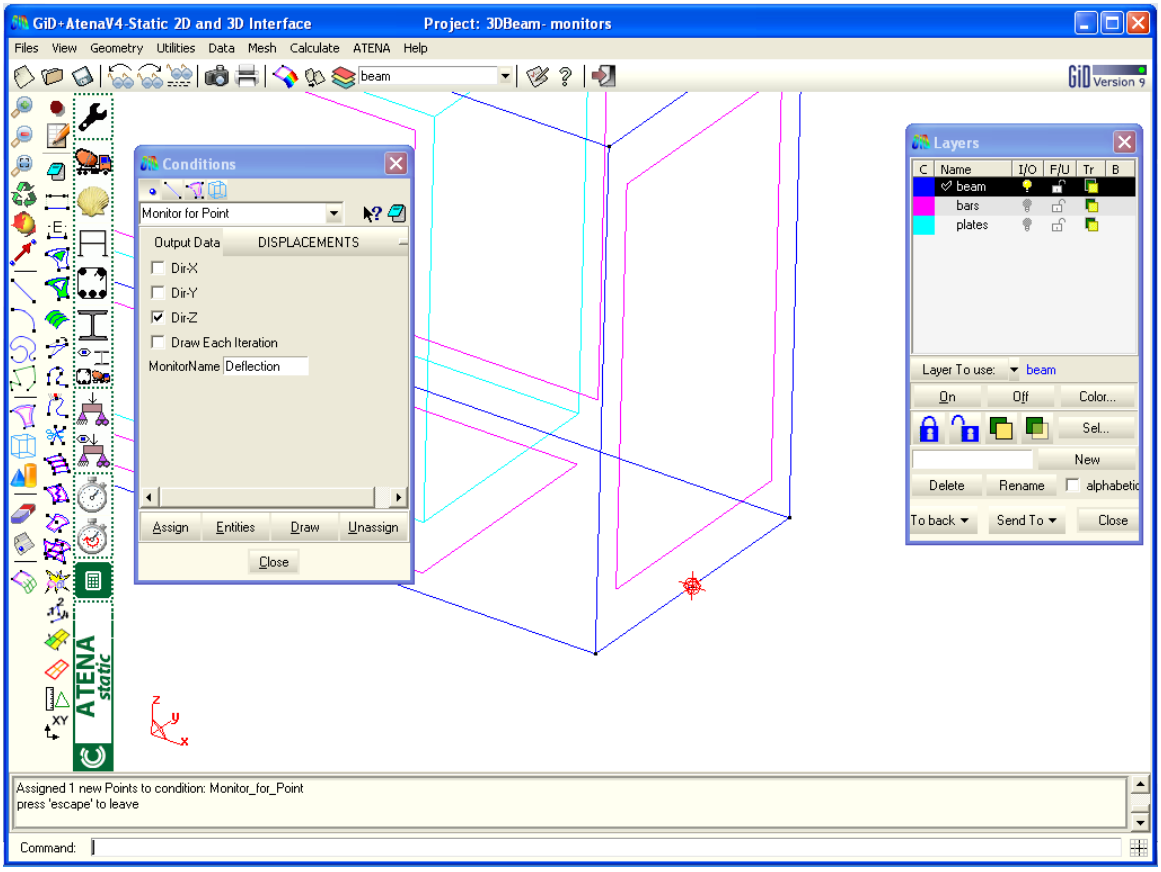


Figure 113: The second monitor condition

All boundary condition should be defined. For control it is recommended to display boundary condition. It can be done by clicking on the icon (see Figure 146).

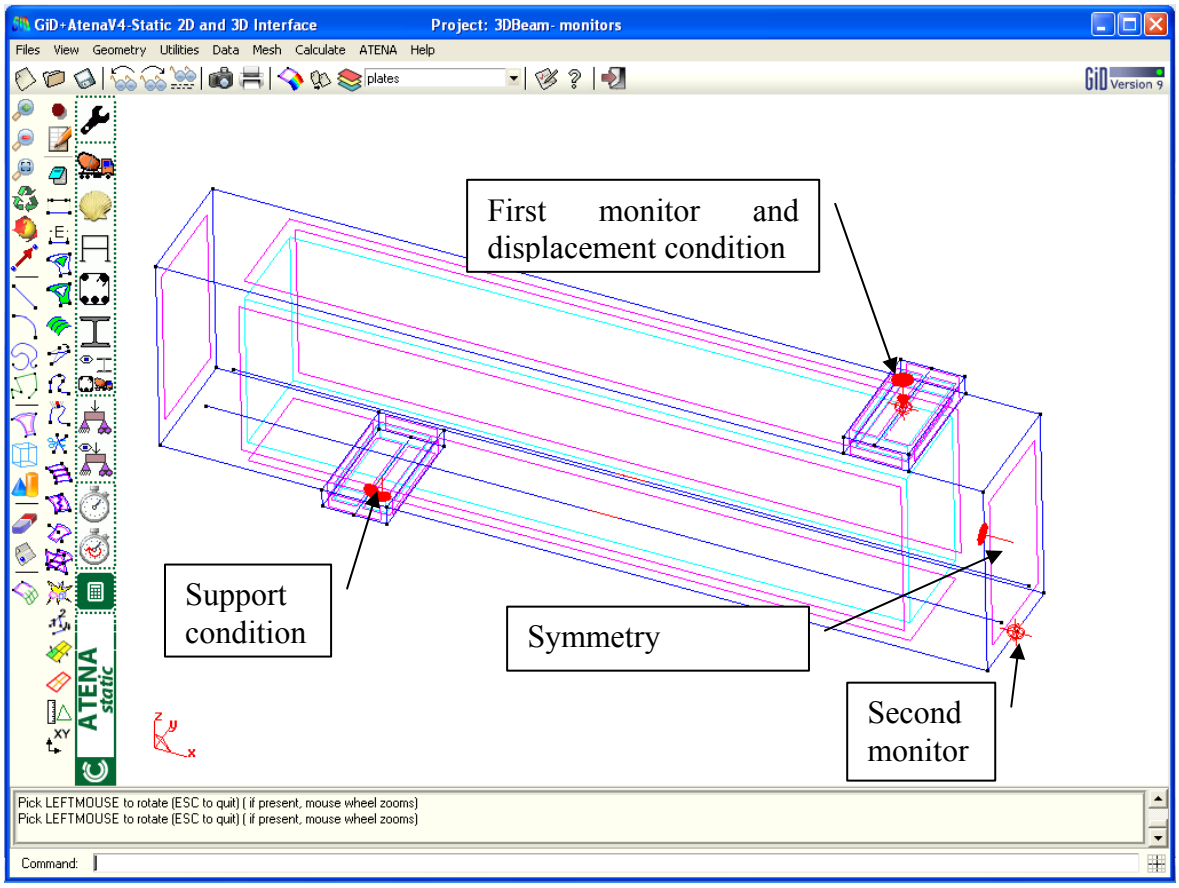

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- monitors
Files View Geometry Utilities Data Mesh Calculate ATENA Help
plates
First monitor and displacement condition
Support condition
Symmetry
Second monitor
ATENA static
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Command:

Figure 114: All boundary conditions

3.5 Intervals – Loading history

This section describes the definition of loading history for the analysis of Leonhardt's shear beam. The loading history consists of intervals. Each interval is divided into load steps. Because in this case the structure is loaded by only one type of force (defined displacement), only one interval will be used. Then this interval will be divided in several steps.

The objective is to keep increasing the load up to failure. Very often before an analysis is started it is difficult to estimate the required loading level that would lead to failure. The maximal load level however, can be often estimated either by simple hand calculation or by performing an initial analysis with a very small load level. Then from the resulting stresses it is possible to estimate how much the load must be increased to fail the structure.

In this example, it is known from the experimental results that the beam should fail at the deflection of about 0.003 m. There is defined a prescribed displacement of 0.0001 m. This means that the predefined displacement should be multiplied approximately 30 times to reach the failure. Base on this assumption, the Load interval will be multiplied by 40. Naturally this interval cannot be applied to the structure in one moment. Therefore it is necessary to divide interval in several load steps. In this case the interval will be divided in 50 load steps.

The loading history can be prescribed by selecting item **`Data | Interval Data`** in the main menu (see Figure 115). After selection of this command the Interval data window will appear and data which should be defined are depicted in the Figure 116.

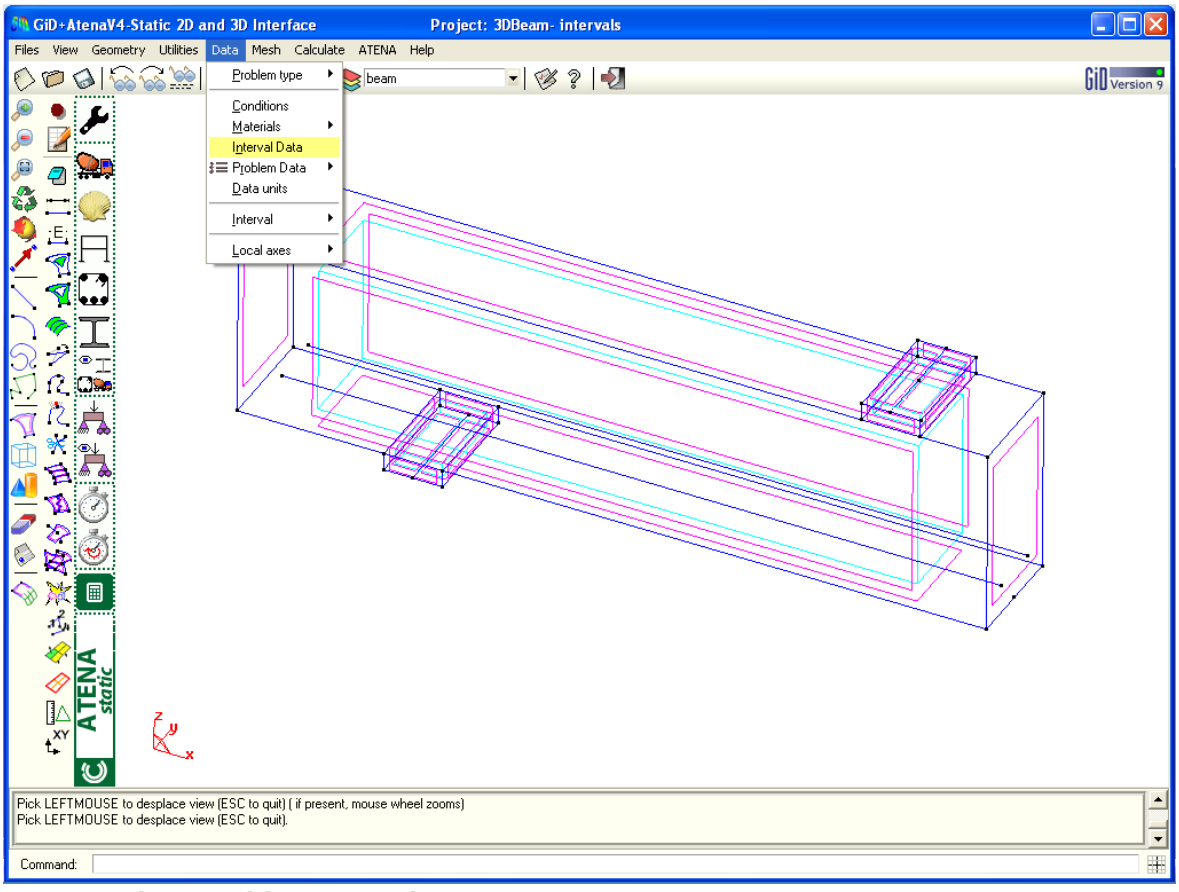


Figure 115: The Interval data command

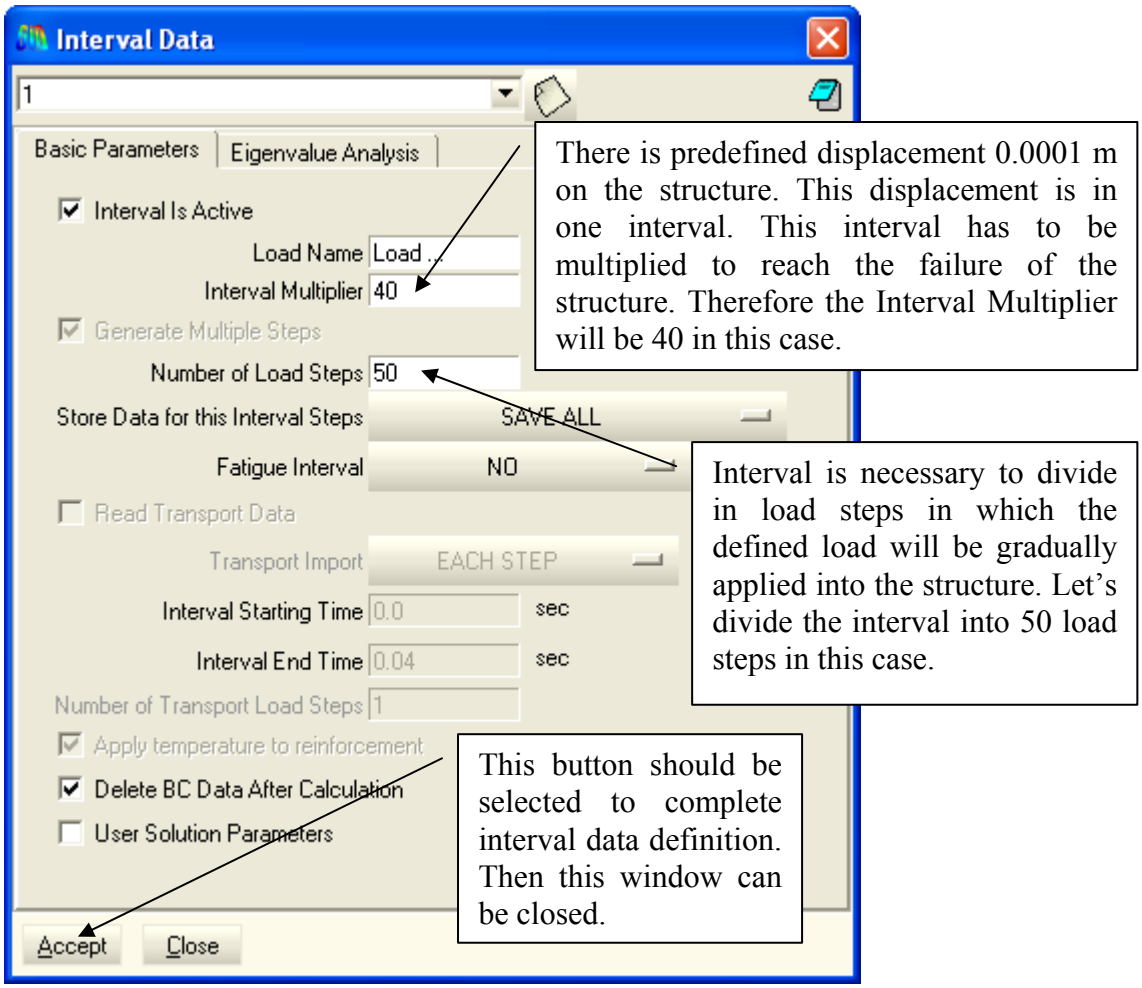

Interval Data
1
Basic Parameters
Eigenvalue Analysis
Interval Is Active
Load Name
Load ...
Interval Multiplier
40
Generate Multiple Steps
Number of Load Steps
50
Store Data for this Interval Steps
SAVE ALL
Fatigue Interval
NO
Read Transport Data
Transport Import
EACH STEP
Interval Starting Time
0.0
sec
Interval End Time
0.04
sec
Number of Transport Load Steps
1
Apply temperature to reinforcement
Delete BC Data After Calculation
User Solution Parameters
Accept
Close
There is predefined displacement 0.0001 m on the structure. This displacement is in one interval. This interval has to be multiplied to reach the failure of the structure. Therefore the Interval Multiplier will be 40 in this case.
Interval is necessary to divide in load steps in which the defined load will be gradually applied into the structure. Let's divide the interval into 50 load steps in this case.
This button should be selected to complete interval data definition. Then this window can be closed.

Figure 116: The contact conditions

Parameter input:

Interval Multiplier: 40

Number of Load Steps: 50

3.6 Mesh generation

The generation of mesh is the last step in pre-processing. Because example in this tutorial should be possible to create in demo version it will be define exact number of elements of mesh. Because of the functionality in the demo version the mesh will be very coarse.

The easiest way of the mesh definition is to use automatic generation. Program will automatically define the smallest suitable mesh. This command can be executed by selecting **`Mesh | Generate mesh`** (see Figure 117) or this option can be activated directly by pressing the key Ctrl and 'g' at the same time. Then the program asks for the definition of the size of the generated mesh (see Figure 118). The default size of the mesh can be used. By the selecting **`Ok`** button the mesh will be generated and the list of elements and nodes of the mesh will appear (see Figure 119).

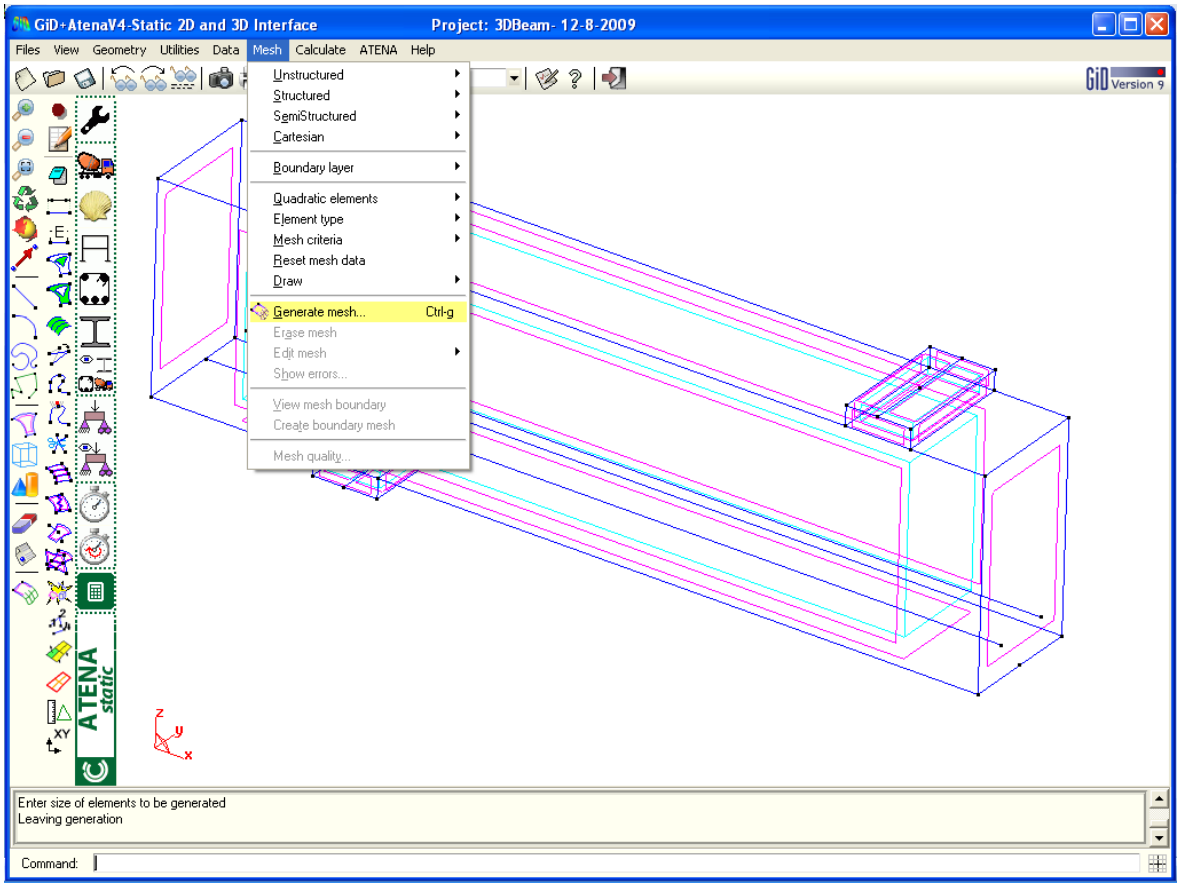


Figure 117: The Generate mesh command

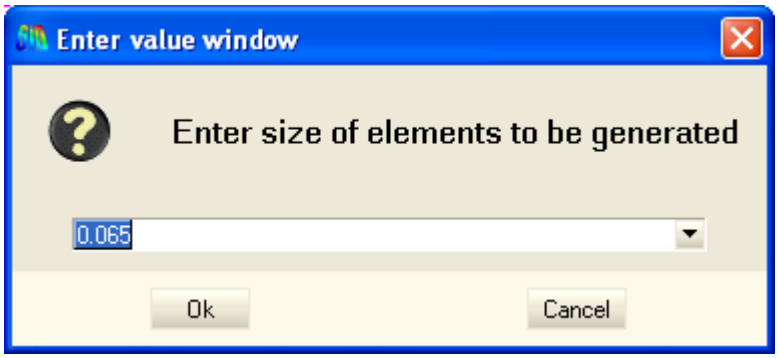


Figure 118: The program offer the size of mesh

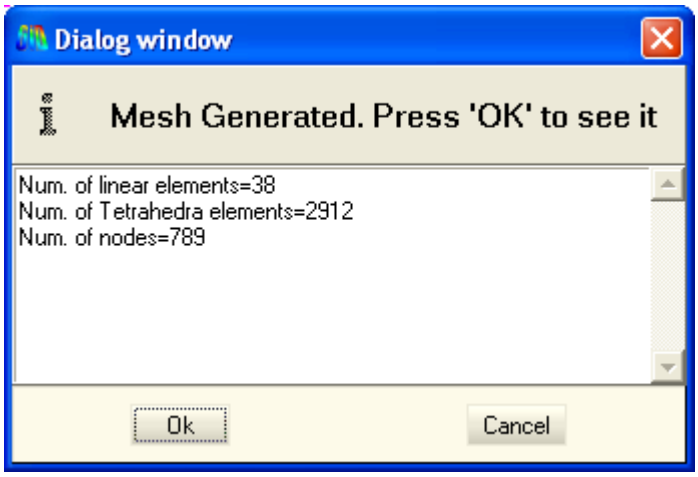


Figure 119: The numbers of elements and node of this geometrical model

The demo version of the **GiD** is limited to 1000 nodes. The example of this tutorial contains 789 nodes. Therefore the automatic sized mesh could be generated (see Figure 120).

But the demo version of **AtenaWin** is limited to 300 elements (see Figure 121). And this mesh contains almost 3000 elements; therefore this mesh will not be functional in **AtenaWin** and the number of element should be decreased.

It can be done by structured mesh which allows controlling numbers of elements. Also in structural analysis it is usually preferred to use brick elements. Therefore in the next steps of the mesh generation the option to create six side brick element will be described.

In this case the structured mesh will be specified only for the beam volume because it is important part of the structure for the structural analysis.

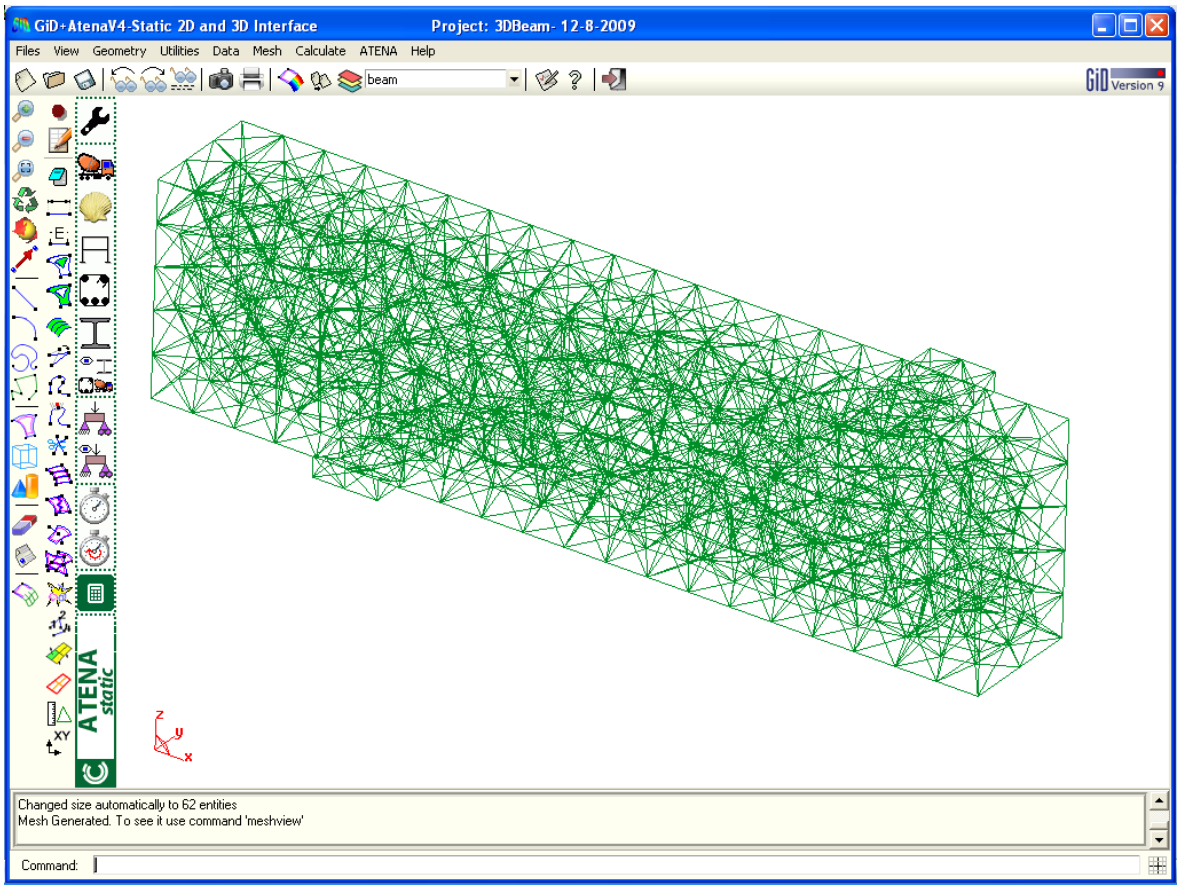

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- 12-8-2009
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Changed size automatically to 62 entities
Mesh Generated. To see it use command 'meshview'
Command:

Figure 120: The generated mesh

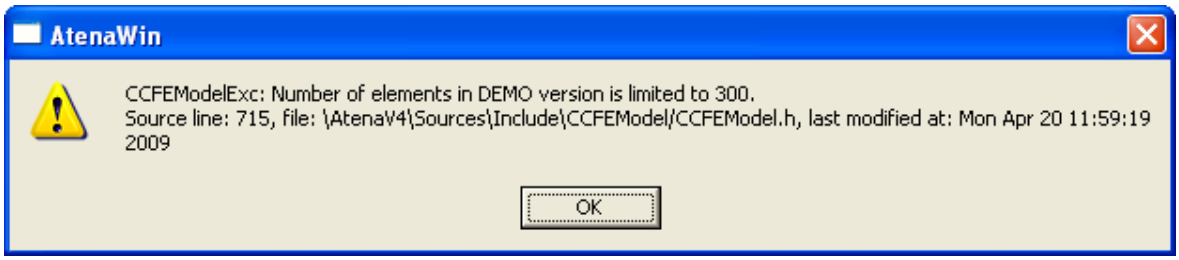

AtenaWin
CCFEModelExc: Number of elements in DEMO version is limited to 300.
Source line: 715, file: \AtenaV4\Sources\Include\CCFEModel/CCFEModel.h, last modified at: Mon Apr 20 11:59:19 2009
OK

Figure 121: The limitation of demo version of ATENA

In this example the beam will have structured brick mesh and steel plates will be meshed with tetrahedral elements.

3.6.1 Structured mesh

Because this example should be possible to create in demo version, the mesh of the beam volume will be structured and limited. The finite element size should be 5 elements over the beam height, 2 elements over the beam width and 16 elements over the beam length.

The structured mesh is done by command **`Mesh | Structured | Volumes`** in the main menu.

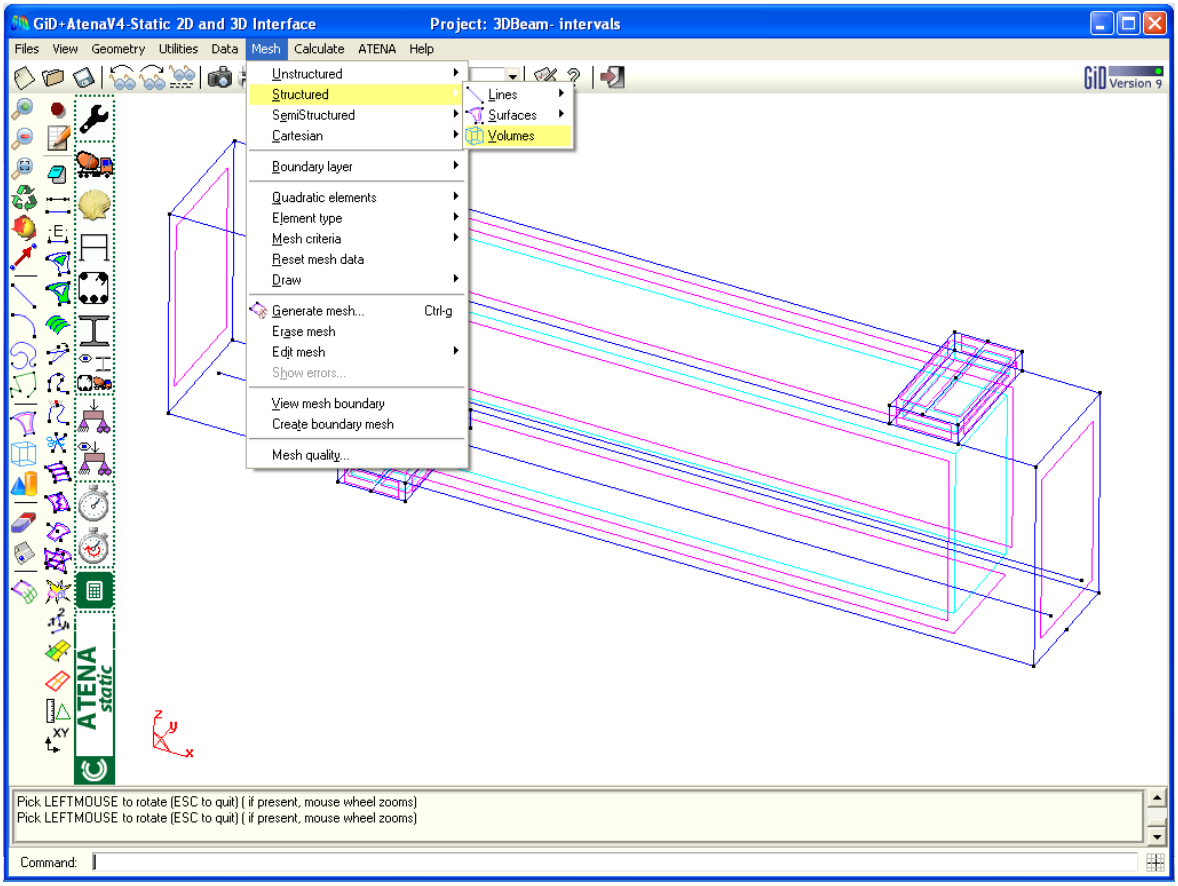


Figure 122: The Structured mesh command

Once this command is executed the volume which should be structured has to be selected (see Figure 123). After the selection the program asks for the number of cells which should be assigned to the lines (see Figure 124).

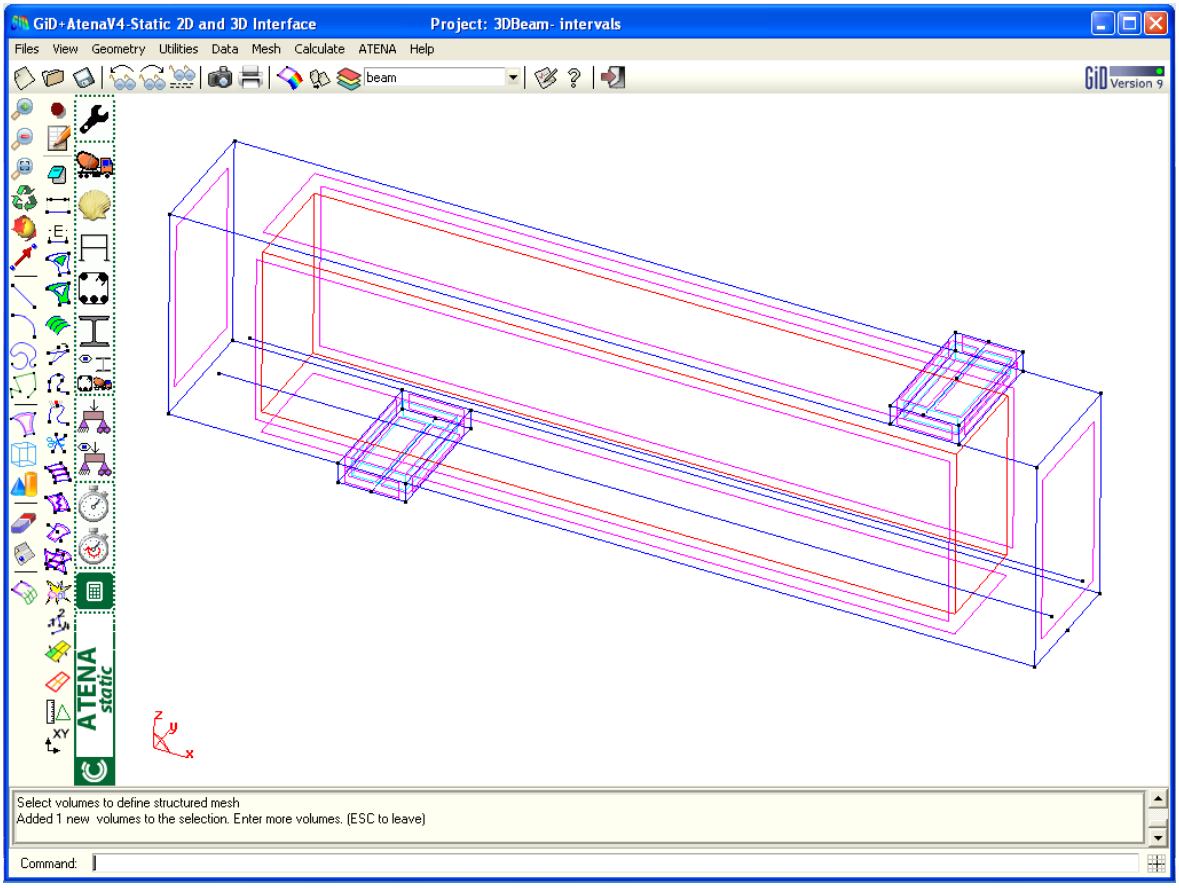

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Select volumes to define structured mesh
Added 1 new volumes to the selection. Enter more volumes. (ESC to leave)
Command:

Figure 123: The selection of the beam volume which should be structured

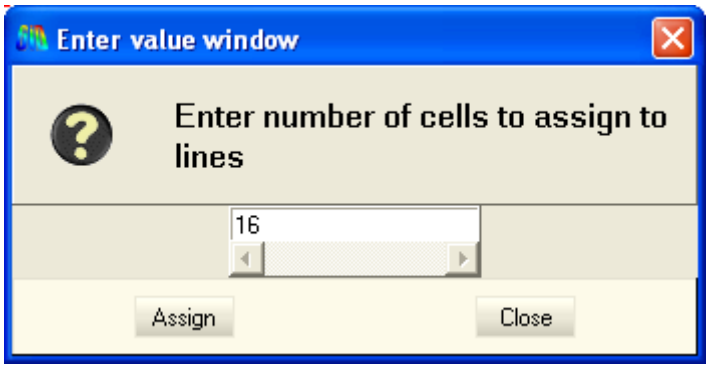


Figure 124: The number of cells for length of the beam

Parameter input:
Enter number of cells to assign to lines: 16

When the number of cells is defined, the button **`Assign`** has to be pressed to select lines which should be structured. The 16 cells will be assigned to the beam length. When the one line of the volume is selected the rest of the lines will be selected automatically (see Figure 125).

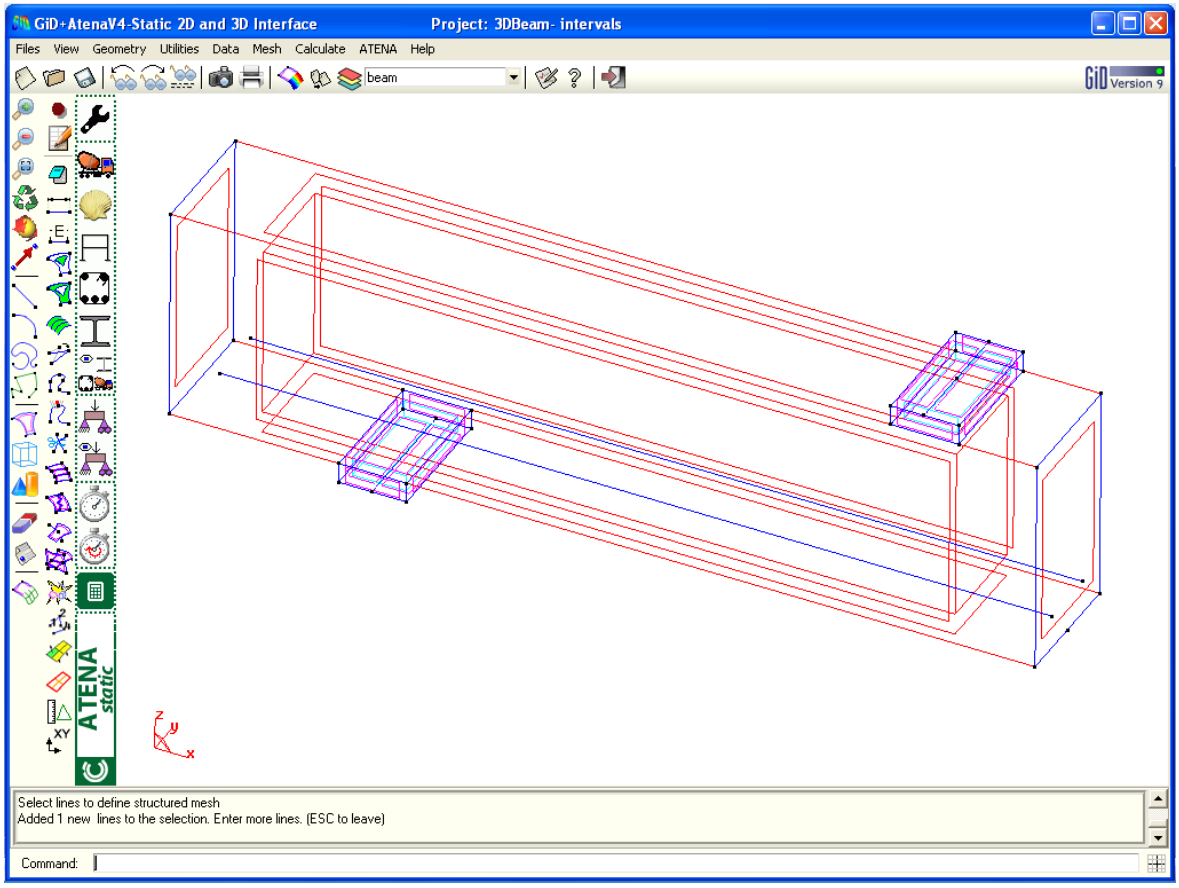

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Select lines to define structured mesh
Added 1 new lines to the selection. Enter more lines. (ESC to leave)
Command:

Figure 125: The selection of the length lines

After the selection the ESC key should be pressed to return to the definition of number of cells. Then the 5 cell should be defined and assigned to the height of the beam (see Figure 126). Then the selection can be done by selecting the button **`Assign`**. The procedure is same like in selection of the length number definition (see Figure 127).

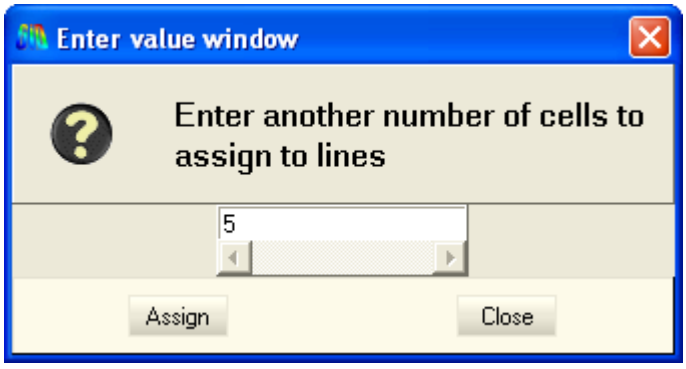


Figure 126: The number of cells for height of the beam

Parameter input:
Enter number of cells to assign to lines: 5

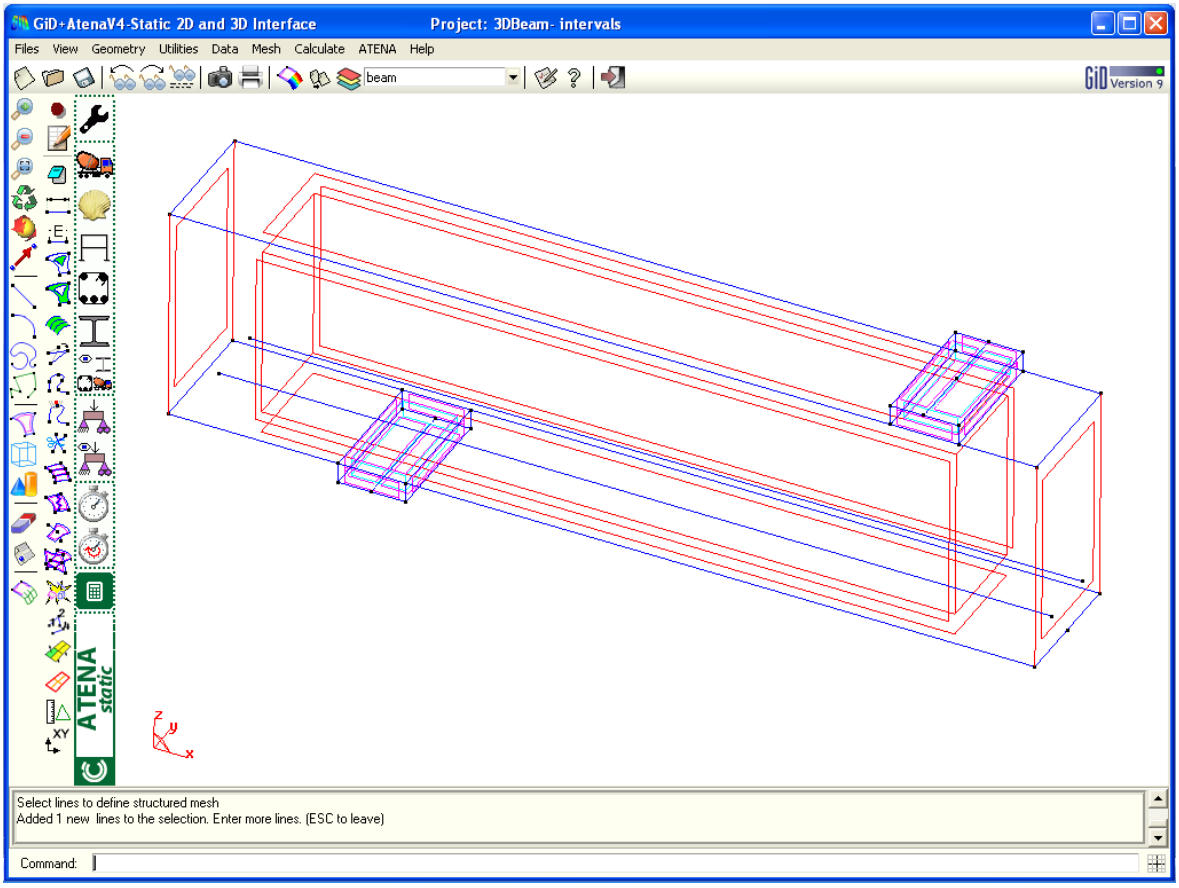

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Select lines to define structured mesh
Added 1 new lines to the selection. Enter more lines. (ESC to leave)
Command:

Figure 127: The selection of the height lines

Next step is to structured width of the beam. The 2 cells should be defined to these lines. Procedure is same like in previous two examples. But when the first line will be selected program will automatically select only other two lines. The last line of the width is divided into two parts. Therefore it is not selected by program and another number of cells has to be assign to these two lines.

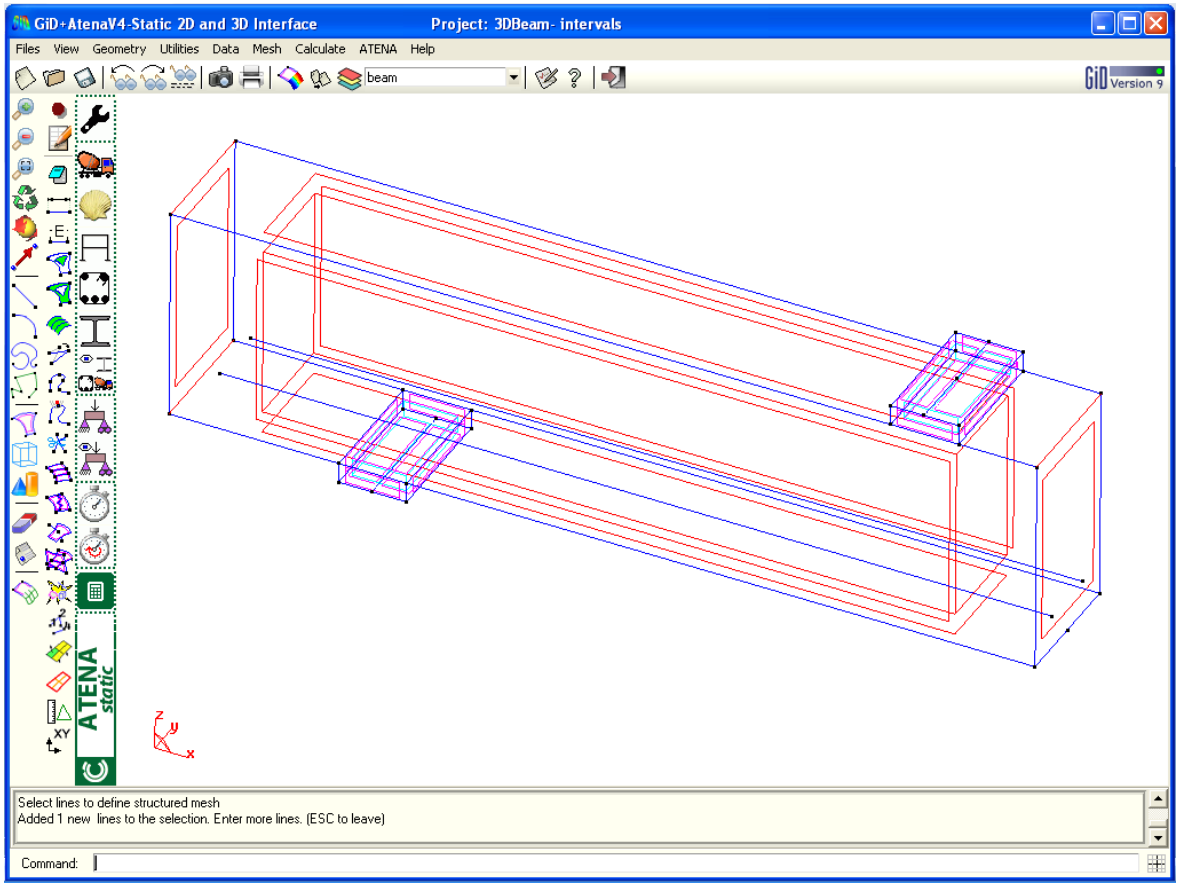

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Select lines to define structured mesh
Added 1 new lines to the selection. Enter more lines. (ESC to leave)
Command:

Figure 128: The selection of the width lines

Last step is to assign 1 cell to the each of the lines of the bottom-right edge of the beam. See Figure 129 and Figure 130.

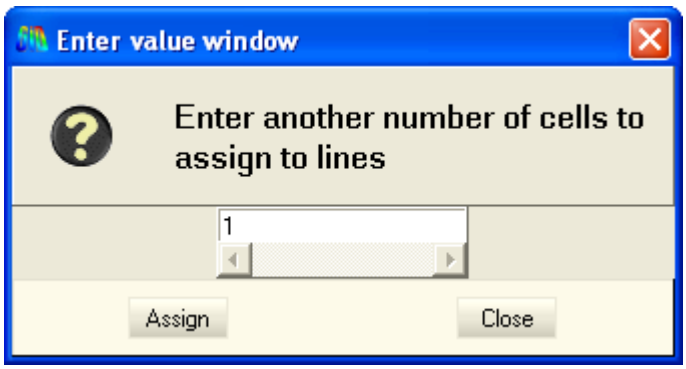


Figure 129: The number of cells for bottom-right edge of the beam

Parameter input:
Enter number of cells to assign to lines: 1

When the volume of the beam is structured the enter value window can be closed by selecting the **Close** button.

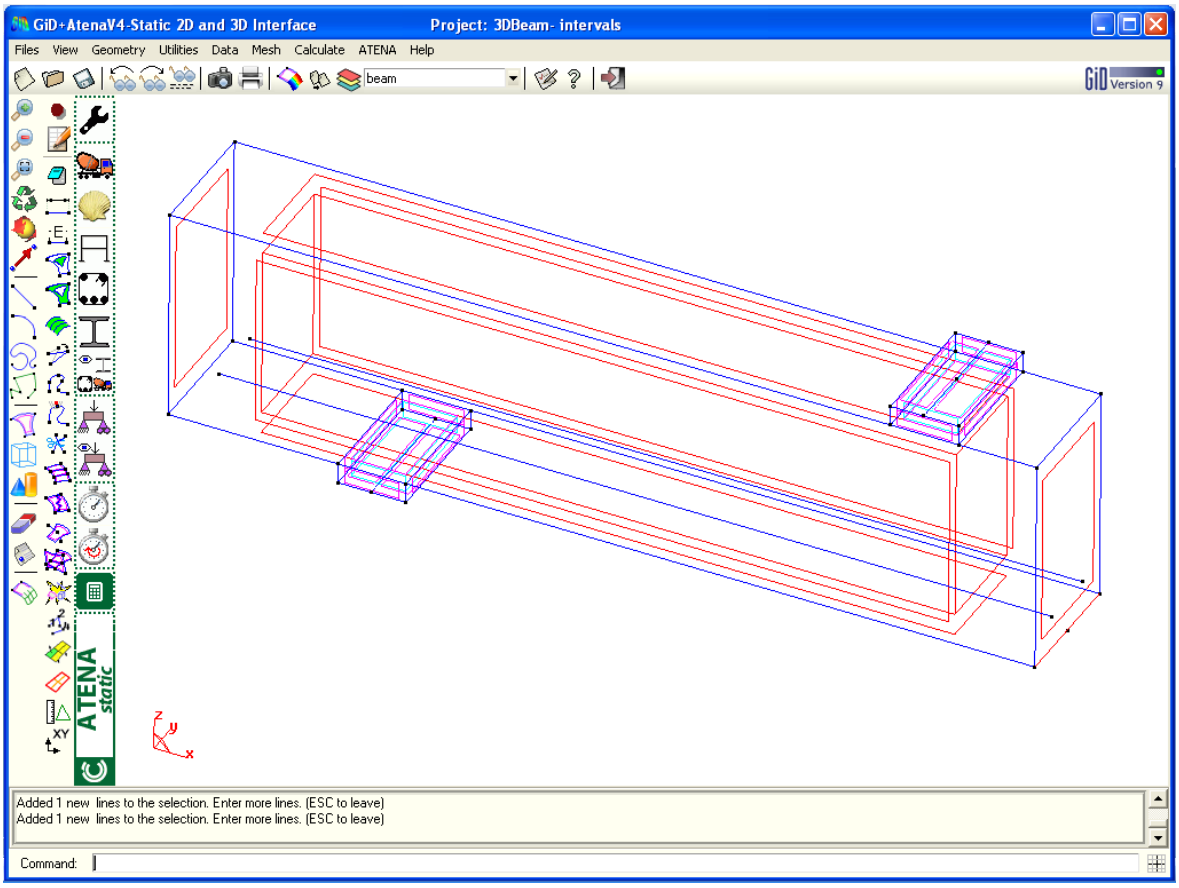

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Added 1 new lines to the selection. Enter more lines. (ESC to leave)
Added 1 new lines to the selection. Enter more lines. (ESC to leave)
Command:

Figure 130: The selection of the bottom-right edge lines

After the structured mesh definition the mesh can be generated. It is done by command **Mesh | Generate mesh** or it can be activated directly by pressing the key Ctrl and 'g' at the same time. After that the enter value window will appear (see Figure 131). There the value 0.065 can be left and the button **Ok** can be pressed. The generation of the mesh will start and then the list of elements will appear. The number of elements can be controlled from that list (see Figure 132). The generation of the mesh is finished by selecting button **Ok** (see Figure 133).

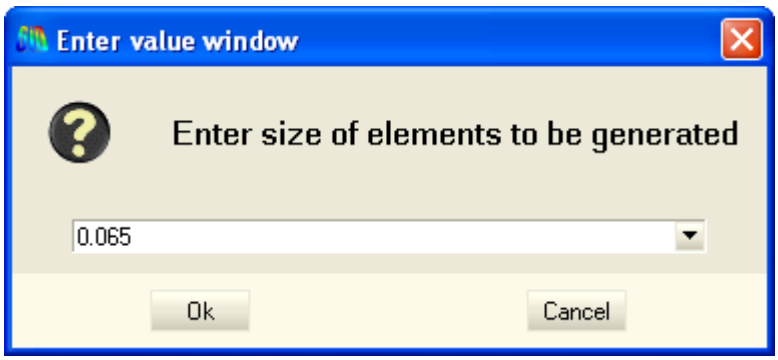


Figure 131: The enter value window

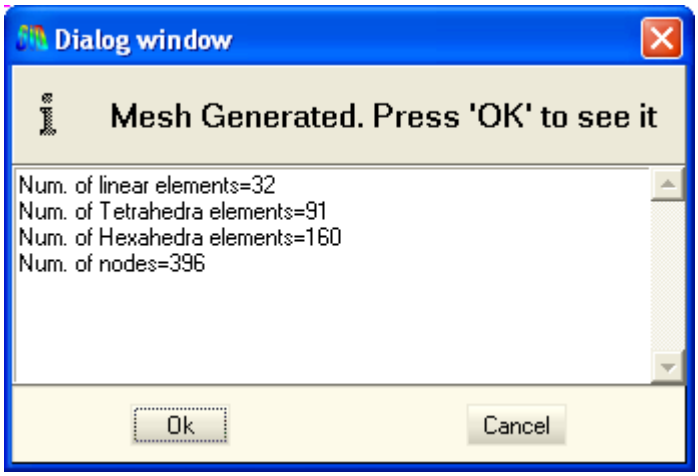


Figure 132: The list of the elements of the mesh

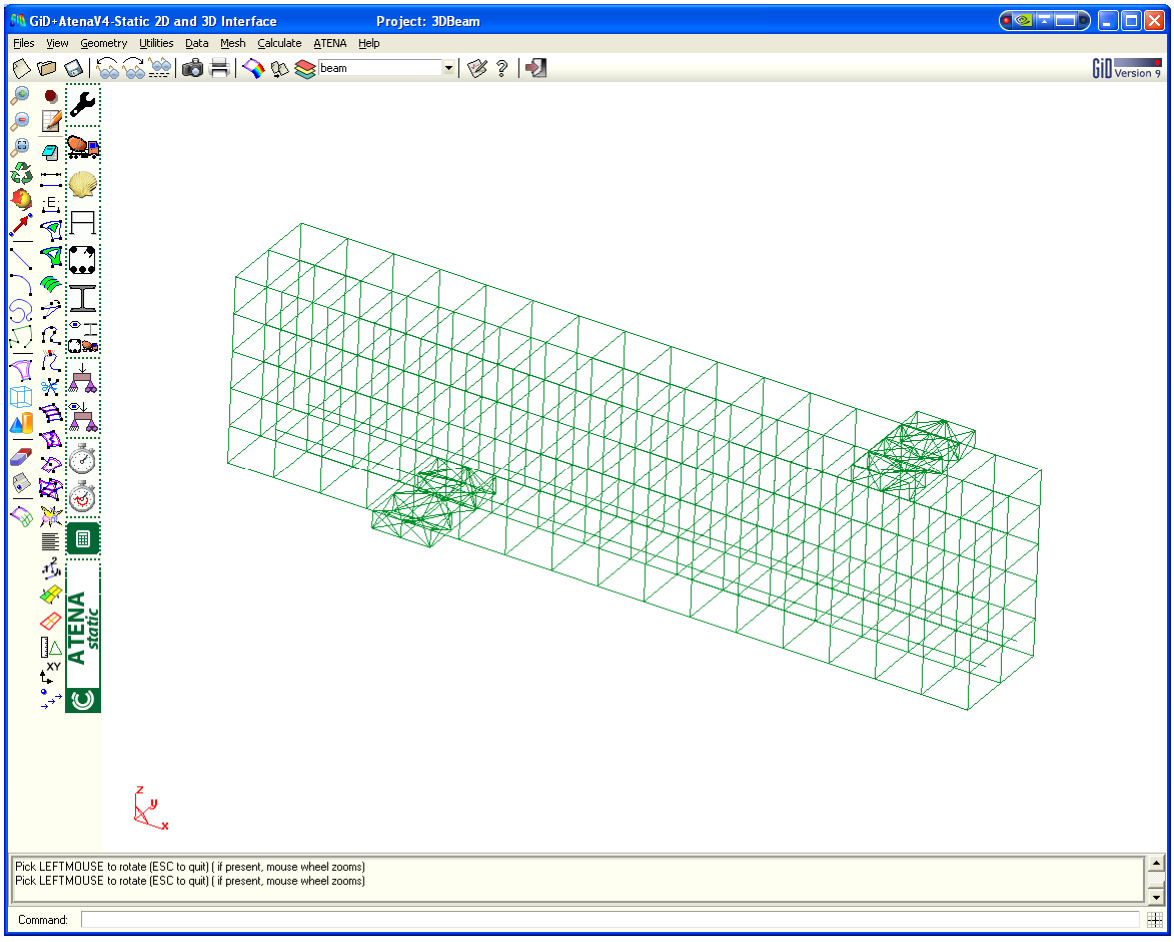

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
ATENA static
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Command:

Figure 133: The generated structured mesh

For better view of the structured mesh the created model can be rendered. It is done by selecting Render in the Mouse menu which appears after clicking on the right-mouse button (see Figure 134).

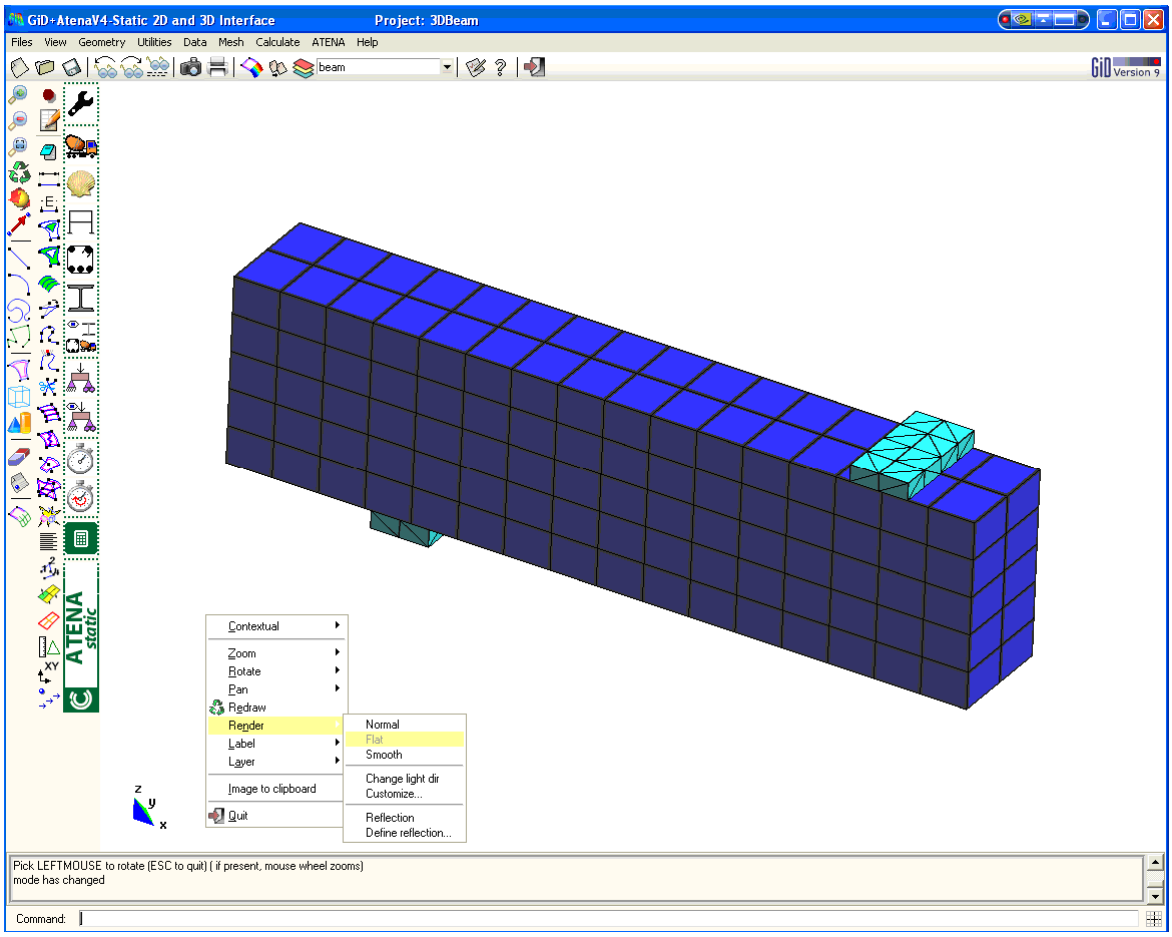


Figure 134: The flat rendered geometrical model

When the mesh is correctly generated the geometrical model definition is finished and calculation can be started. See following chapter 4.

4. FE NON-LINEAR ANALYSIS

This chapter describes the process of running a non-linear analysis of the Leonhardt beam using the data that have been prepared in the previous sections of the tutorial.

The finite element analysis is started by the clicking on the icon or by the using of command **`Calculate | Calculate`**. After selecting this command, the program will start to generate the input files for each step of the non-linear analysis. This process is indicated by the dialog box (see Figure 135). And then the AtenaWin window will appear and analysis will be in progress (see Figure 136).

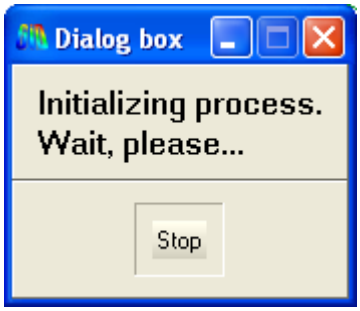


Figure 135: The initializing of the analysis

If the creation of the geometrical model and definition of the boundary conditions were done right, the static analysis should be finished in one minute. Then in the Geometry window it is possible to see that the loading steel plate was moved (see Figure 137) by the applied loads and therefore the structure could not be calculated correctly. It is because there is no connection between concrete beam and steel plates. Program does not automatically detect possible contact between volumes. Contacts have to be added manually by boundary special conditions.

Therefore, it is necessary to return back to the **GiD** graphical interface and defined fixed contacts. **AtenaWin** can be simply closed without any savings of data and then it is necessary to return back to the **GiD** graphical interface and define fixed contacts.

In this tutorial the geometry is created by three separated volumes and therefore contacts have to be added. If the corresponding surfaces of the steel plates would be parts of the geometry of the beam, all parts of the structure would be connected and no special condition would have to be defined.

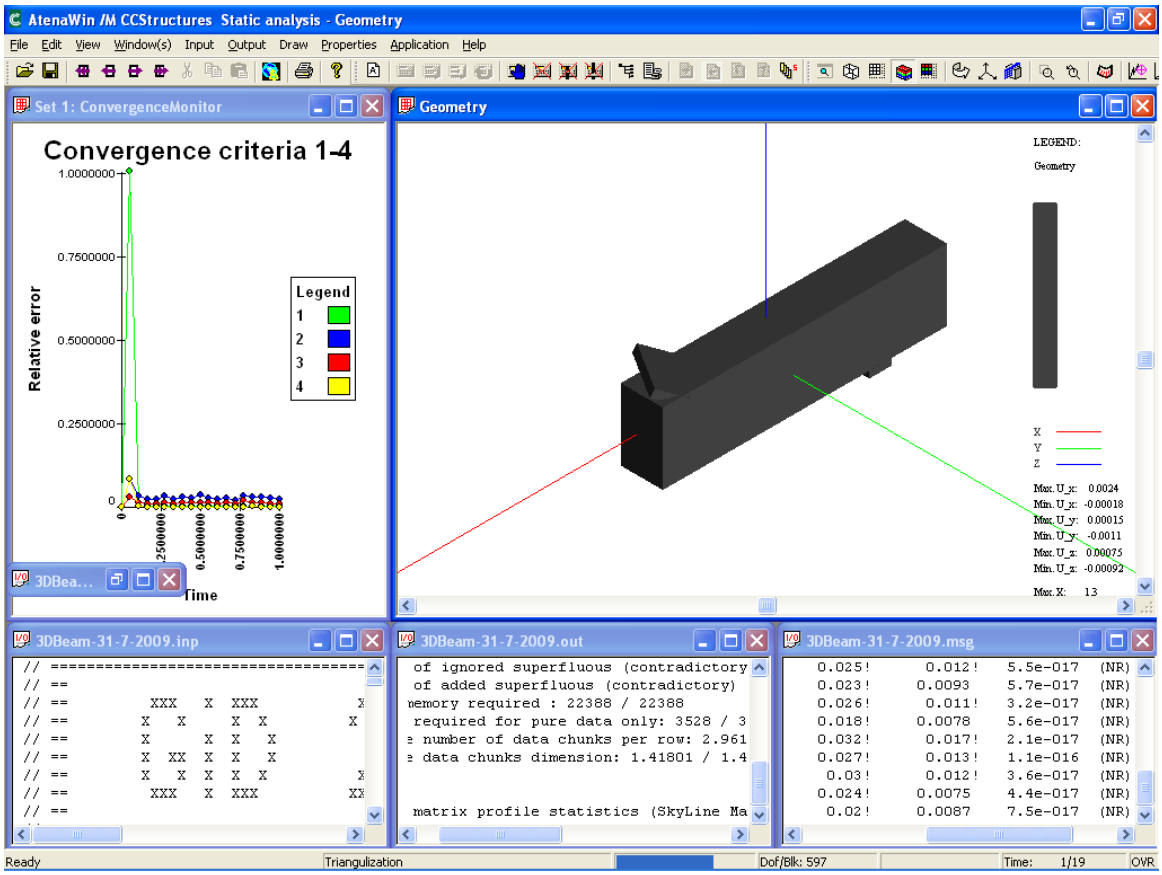


Figure 136: The AtenaWin interface window

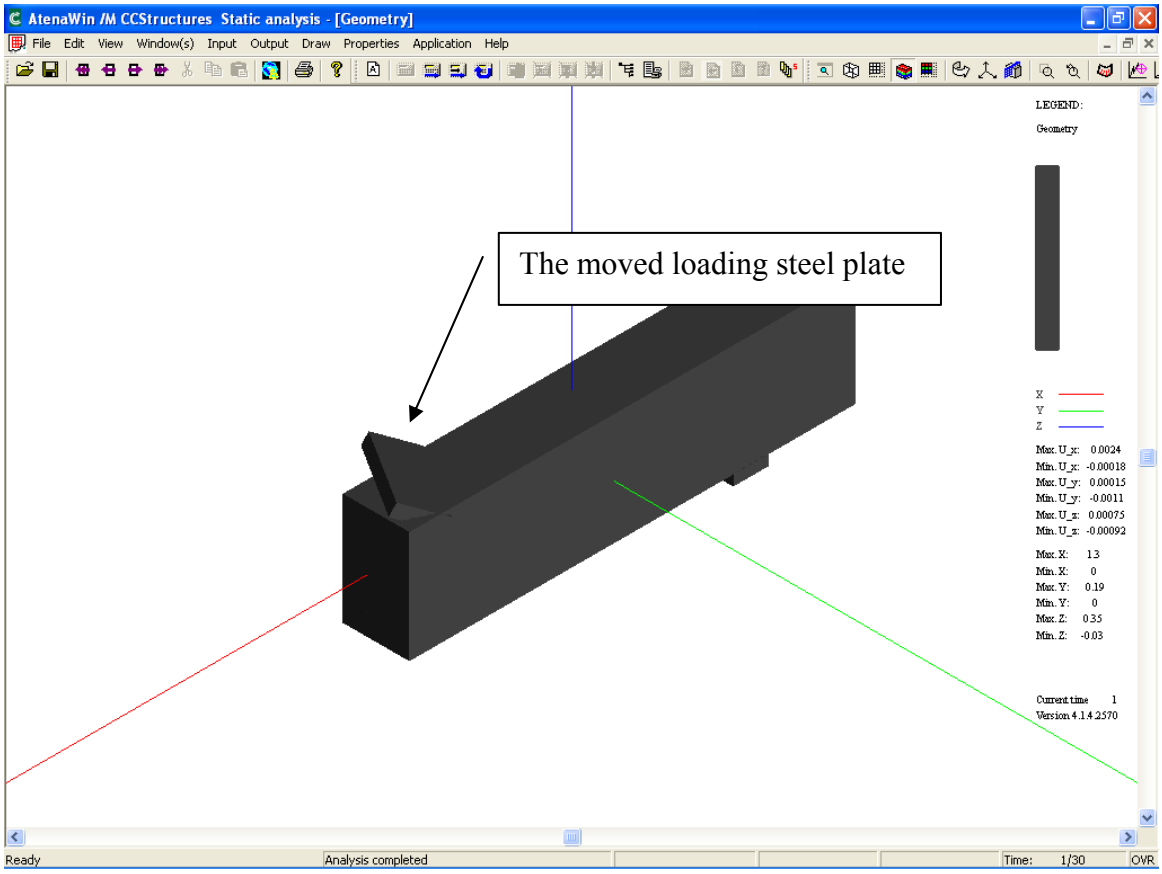


Figure 137: The moved loading steel plate.

There should be Info window in the GiD (see Figure 138). This informative window can be closed and the definition of the fixed contacts can be started (see 4.1).

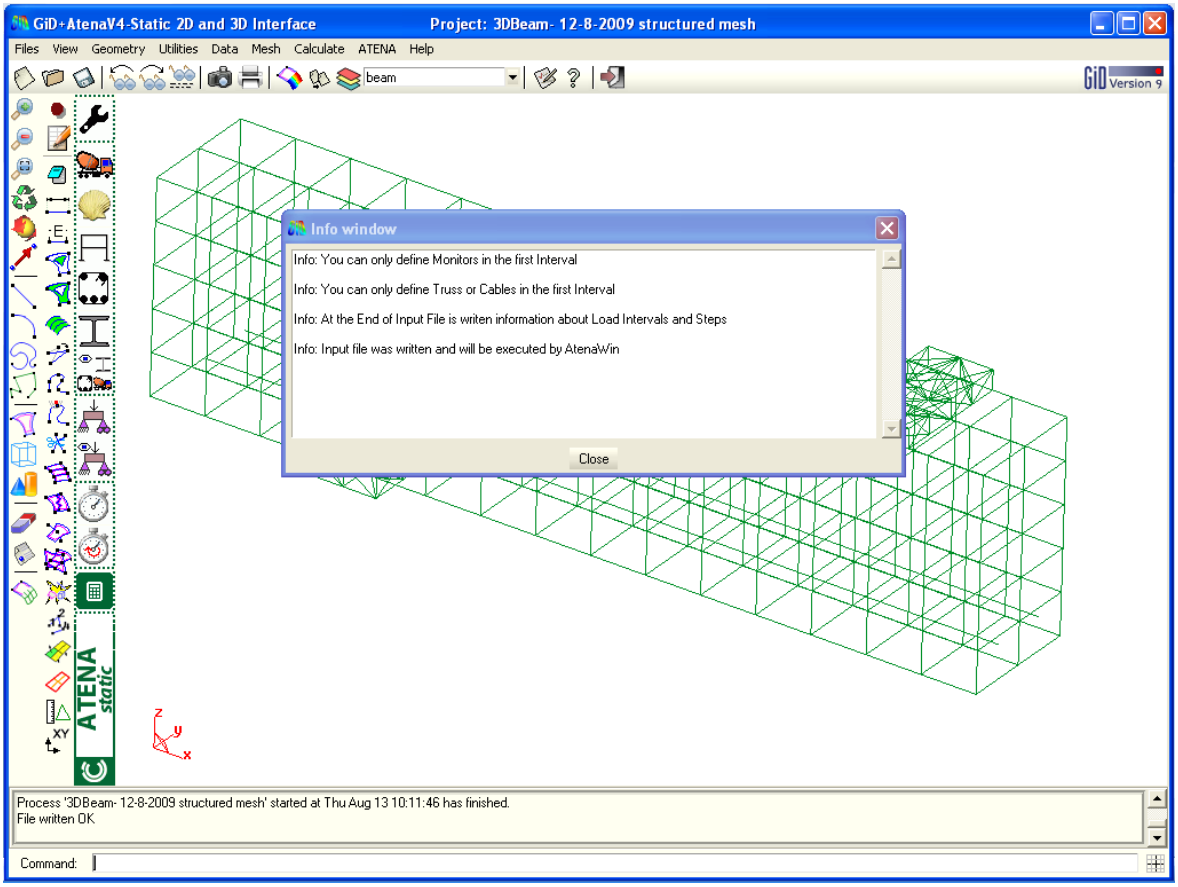

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- 12-8-2009 structured mesh
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
Info window
Info: You can only define Monitors in the first Interval
Info: You can only define Truss or Cables in the first Interval
Info: At the End of Input File is writen information about Load Intervals and Steps
Info: Input file was written and will be executed by AtenaWin
Close
ATENA static
Process '3DBeam- 12-8-2009 structured mesh' started at Thu Aug 13 10:11:46 has finished.
File written OK
Command:

Figure 138: The GiD interface after analysis

4.1 Missing contacts

The geometry is composed from three 3D regions – concrete beam and two steel plates. These regions should be connected. However in this example there is no connection yet. Therefore the contact condition has to be added.

Contact condition distinguishes Master and Slave condition. In this case the beam will be master and plates will be slaves. Therefore four contact conditions have to be added – two master conditions on beam (top and bottom) and two slave conditions on plates (top and bottom).

Before condition definition, by the clicking on the icon the mesh will be hidden and geometry will be displayed. It can be also done by selecting command **`Geometry | View geometry`** in the main menu.

4.1.1 Master Top beam condition

Conditions command can be executed by the selection of the icon or by the selecting the command **Data | Conditions** in the main menu. The contact condition definition for master top beam is depicted in the Figure 139.

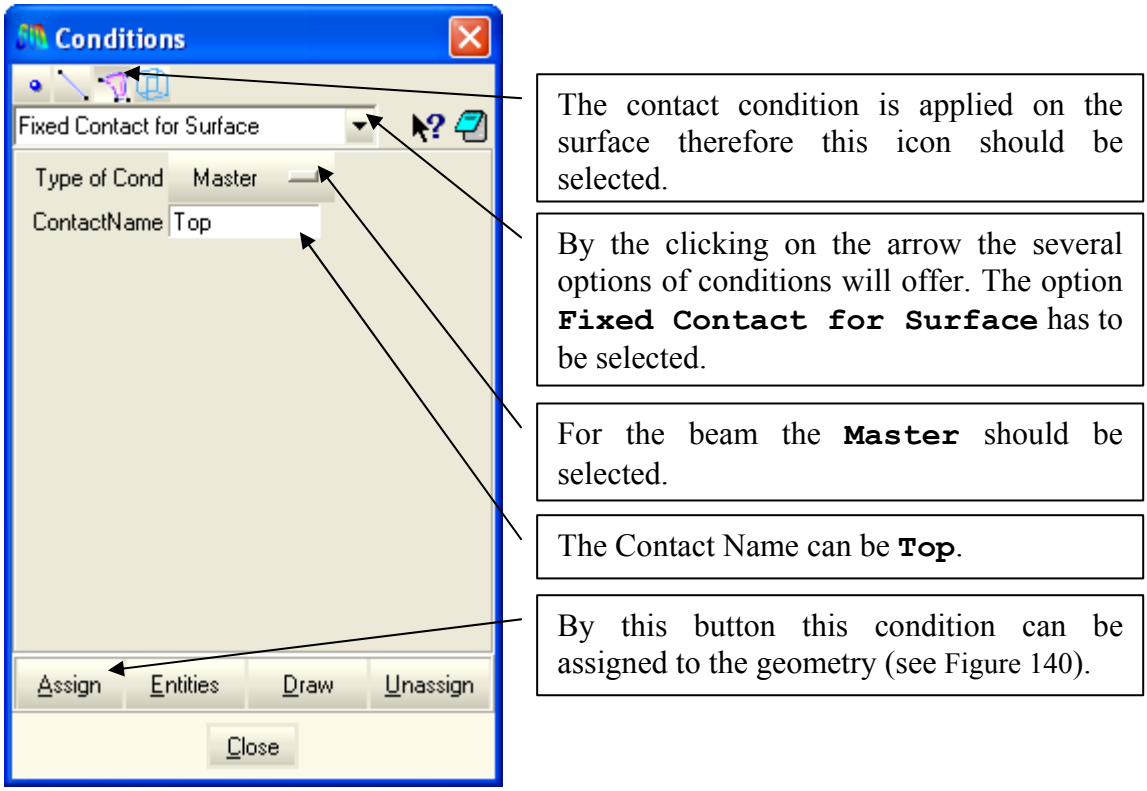

Conditions
Fixed Contact for Surface
Type of Cond
Master
ContactName
Top
Assign
Entities
Draw
Unassign
Close
The contact condition is applied on the surface therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will offer. The option Fixed Contact for Surface has to be selected.
For the beam the Master should be selected.
The Contact Name can be Top.
By this button this condition can be assigned to the geometry (see Figure 140).

Figure 139: The master top beam contact condition

Parameter input:
Fixed Contact for Surface
Type of Cond: MASTER
Contact Name: Top

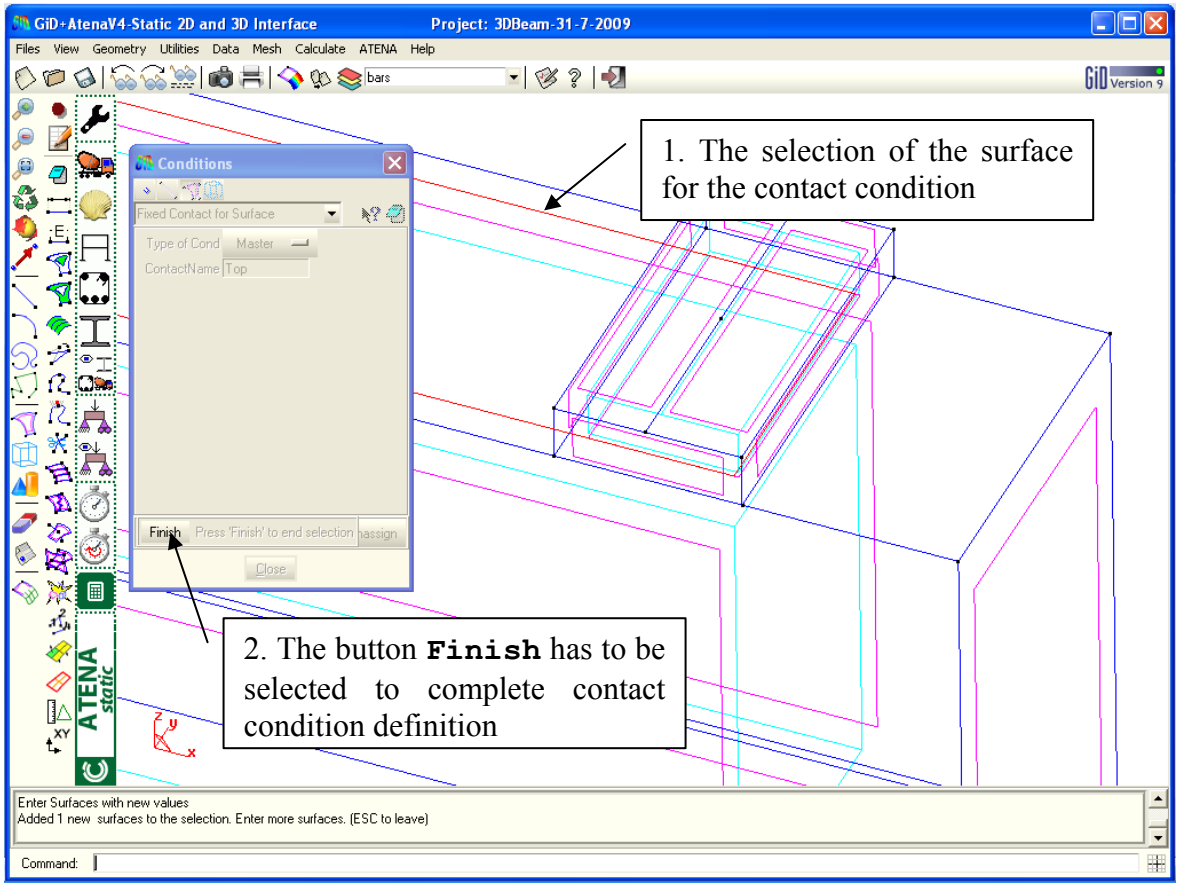

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam-31-7-2009
Files View Geometry Utilities Data Mesh Calculate ATENA Help
bars
GiD Version 9
Conditions
Fixed Contact for Surface
Type of Cond Master
ContactName Top
Finish
Press 'Finish' to end selection
Close
1. The selection of the surface for the contact condition
2. The button Finish has to be selected to complete contact condition definition
ATENA static
Enter Surfaces with new values
Added 1 new surfaces to the selection. Enter more surfaces. (ESC to leave)
Command:

Figure 140: The selection of the surface for the master top beam contact condition

Next the command draw condition has to be selected to display defined condition. The button **Draw** should be selected in the bottom of the Conditions window. After clicking on that button several options will appear (see Figure 141). For example the **Colors** can be selected and the master contact condition will be drawn (see Figure 142).

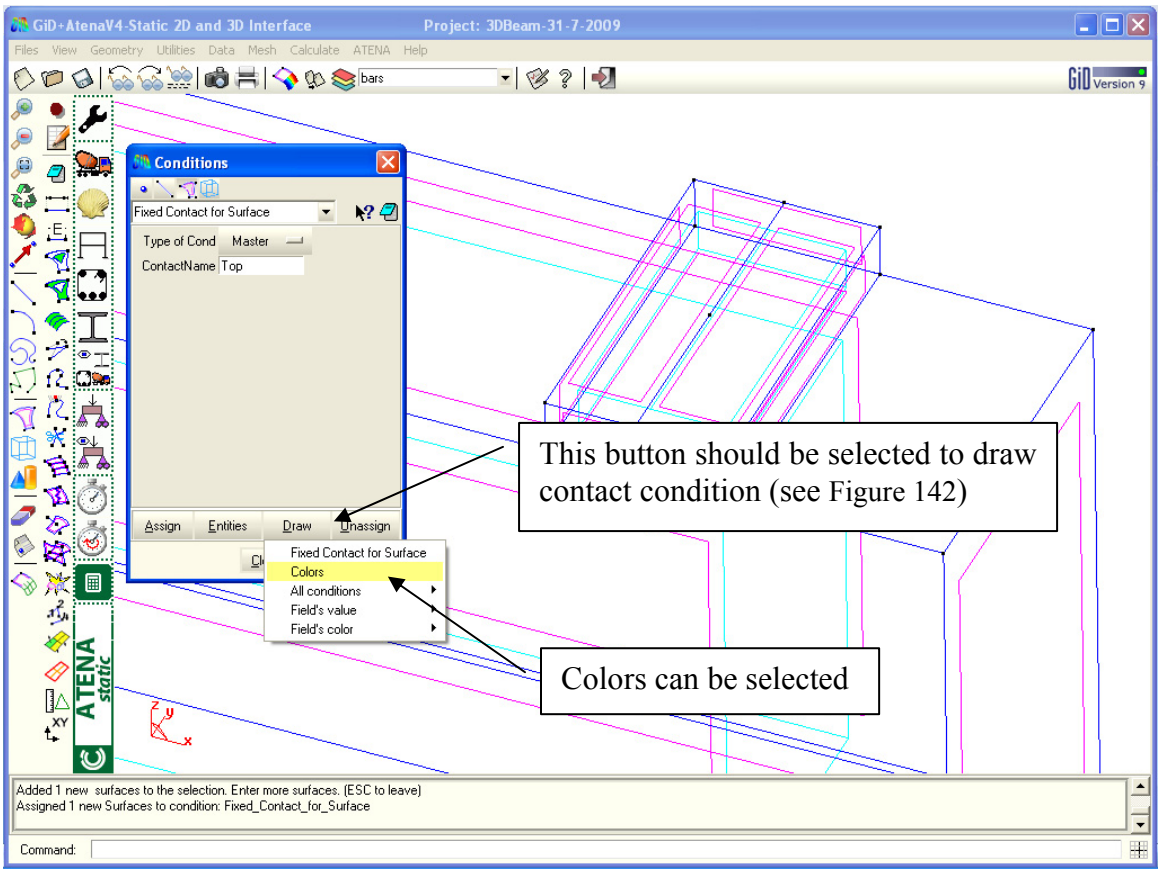


Figure 141: The draw coloured contact condition command

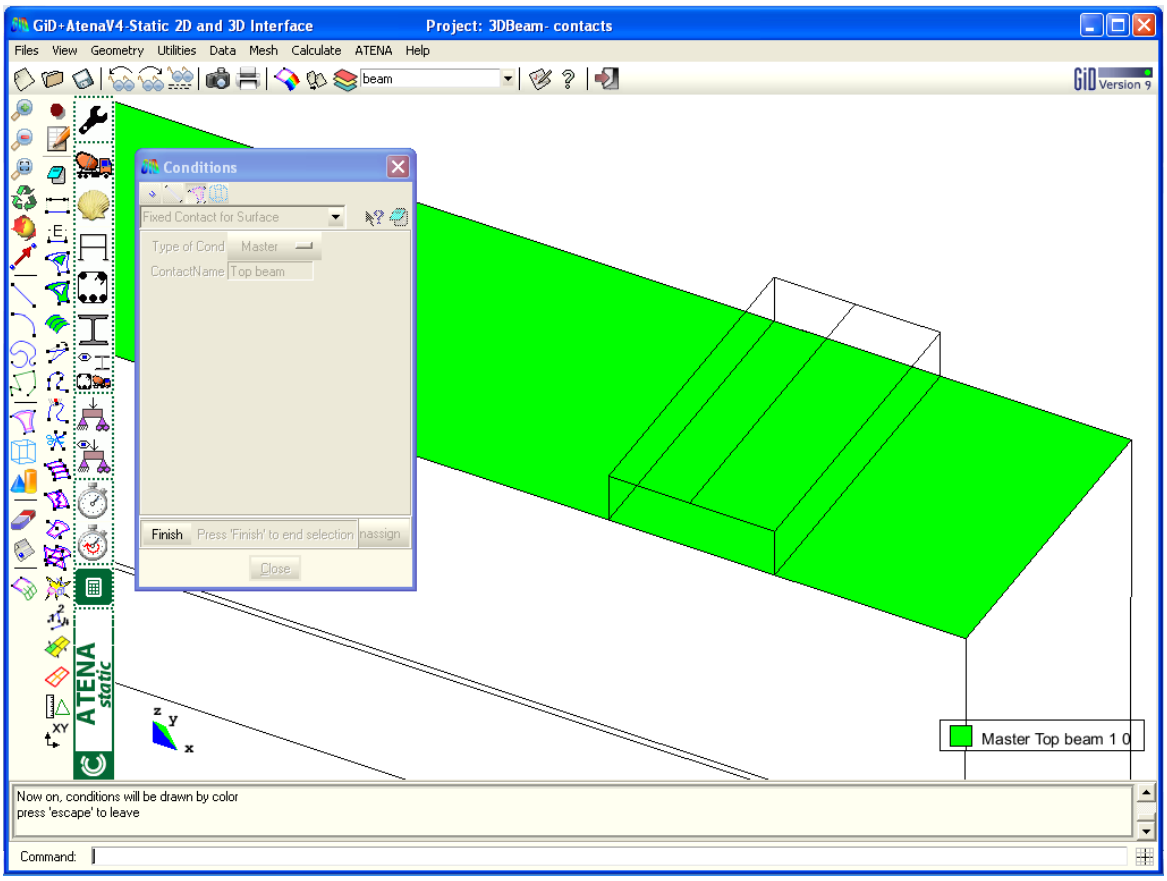


Figure 142: The Master Top beam condition

4.1.2 Slave Top plate condition

Conditions command can be executed by the selection of the icon or by the selecting the command **Data | Conditions** in the main menu. The contact condition definition for master top beam is depicted in the Figure 143.

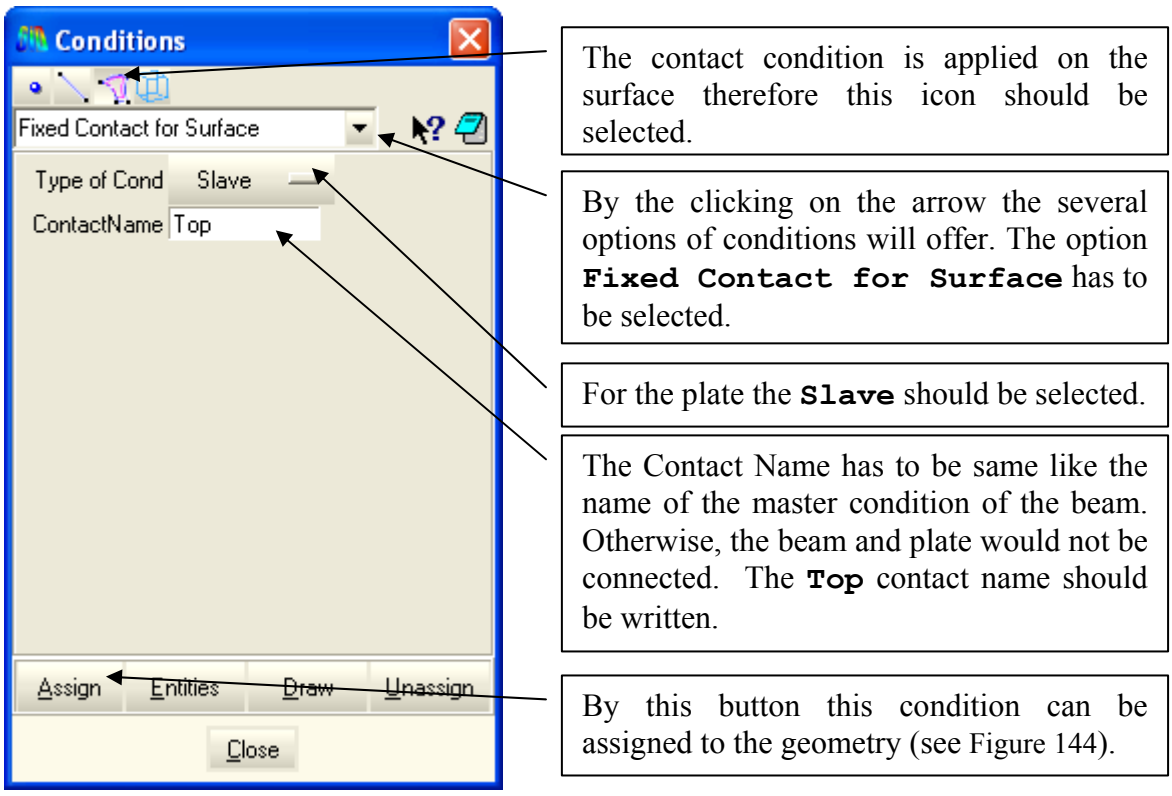

Conditions
Fixed Contact for Surface
Type of Cond
Slave
ContactName
Top
Assign
Entities
Draw
Unassign
Close
The contact condition is applied on the surface therefore this icon should be selected.
By the clicking on the arrow the several options of conditions will offer. The option Fixed Contact for Surface has to be selected.
For the plate the Slave should be selected.
The Contact Name has to be same like the name of the master condition of the beam. Otherwise, the beam and plate would not be connected. The Top contact name should be written.
By this button this condition can be assigned to the geometry (see Figure 144).

Figure 143: The slave top plate contact condition

Parameter input:
Fixed Contact for Surface
Type of Cond: SLAVE
Contact Name: Top

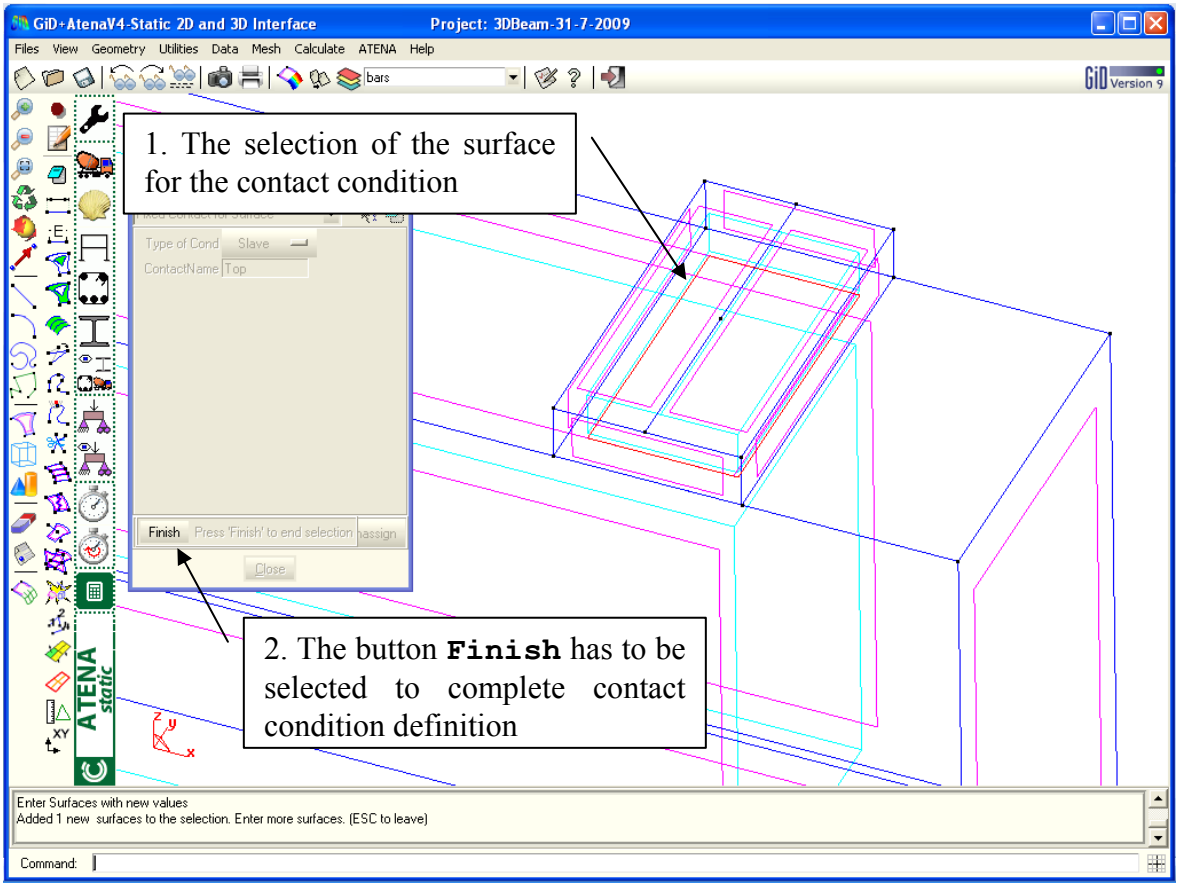

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam-31-7-2009
Files View Geometry Utilities Data Mesh Calculate ATENA Help
bars
GiD Version 9
1. The selection of the surface for the contact condition
Type of Cond Slave
ContactName Top
Finish
Press 'Finish' to end selection
Close
2. The button Finish has to be selected to complete contact condition definition
ATENA static
Enter Surfaces with new values
Added 1 new surfaces to the selection. Enter more surfaces. (ESC to leave)
Command:

Figure 144: The selection of the surface for the slave top plate contact condition

4.1.3 Master bottom beam and Slave bottom plate conditions

The bottom conditions will be done by the same procedure like in the case of top contact conditions. Only the name has to be different. It is recommended to use contact name Bottom. The Figure 145 shows the right definition of bottom contact conditions.

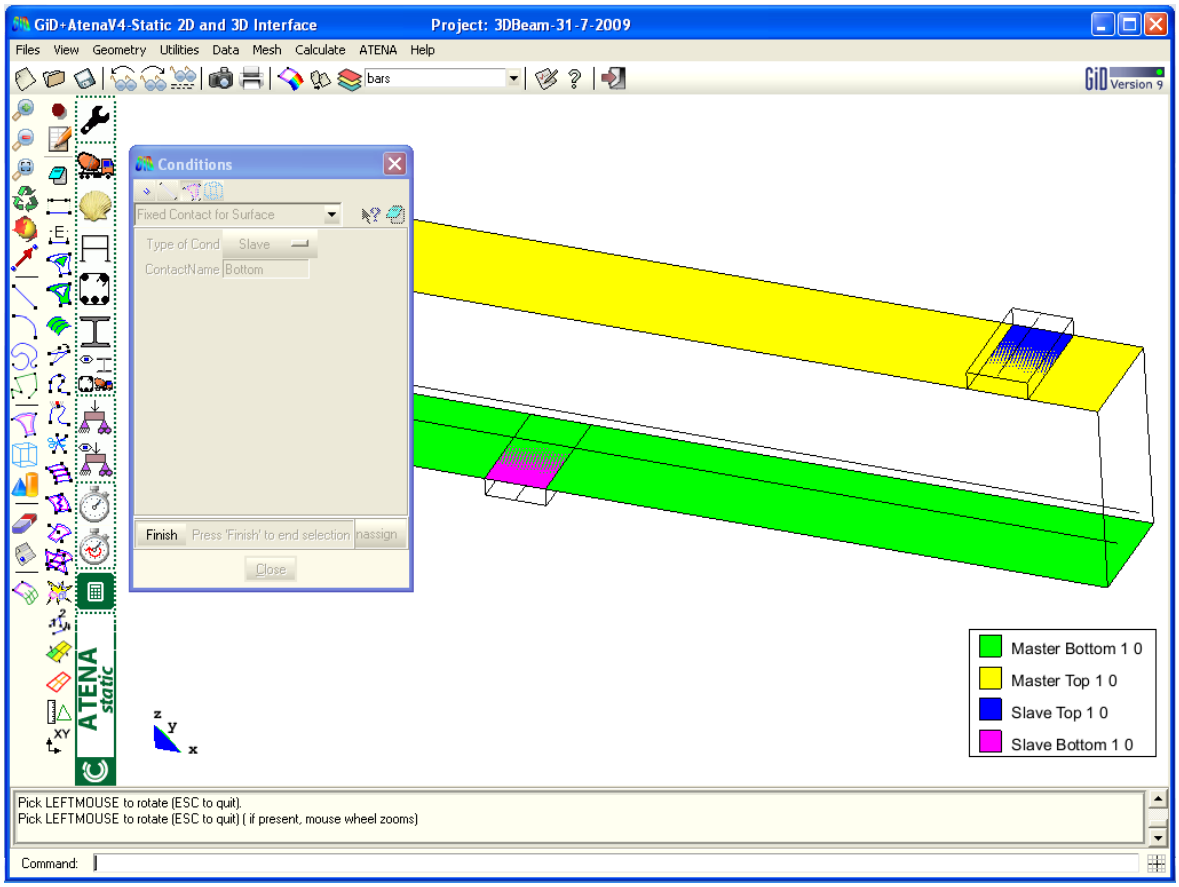

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam-31-7-2009
Files View Geometry Utilities Data Mesh Calculate ATENA Help
bars
GiD Version 9
Conditions
Fixed Contact for Surface
Type of Cond Slave
ContactName Bottom
Finish Press 'Finish' to end selection nassign
Close
ATENA static
Master Bottom 1 0
Master Top 1 0
Slave Top 1 0
Slave Bottom 1 0
Pick LEFTMOUSE to rotate (ESC to quit).
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Command:

Figure 145: The contact conditions

B the clicking in the icon all boundary condition can be displayed. It is good control if all conditions were properly defined (see Figure 146).

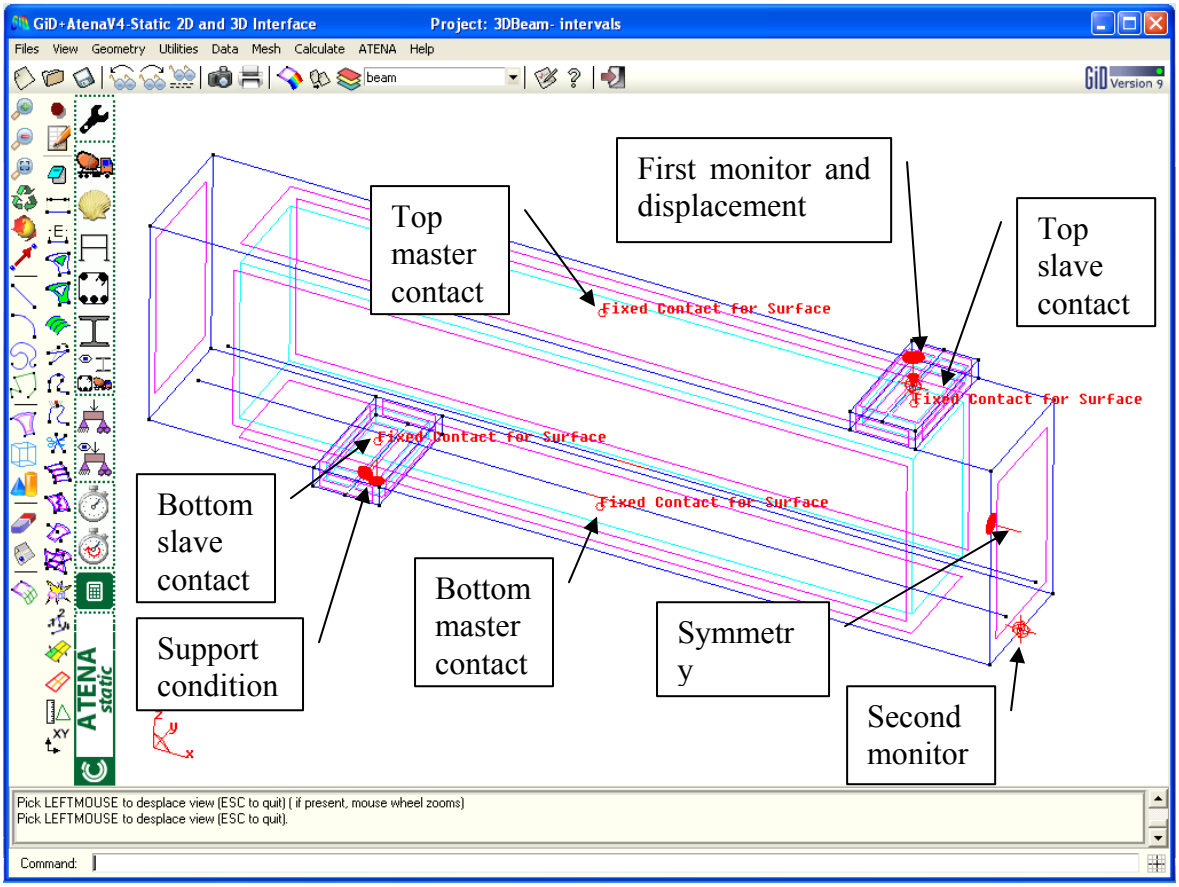

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- intervals
Files View Geometry Utilities Data Mesh Calculate ATENA Help
beam
GiD Version 9
First monitor and displacement
Top master contact
Top slave contact
Fixed Contact for Surface
Fixed Contact for Surface
Fixed Contact for Surface
Fixed Contact for Surface
Bottom slave contact
Support condition
Bottom master contact
Symmetr y
Second monitor
ATENA static
Pick LEFTMOUSE to desplace view (ESC to quit) (if present, mouse wheel zooms)
Pick LEFTMOUSE to desplace view (ESC to quit).
Command:

Figure 146: All boundary conditions

When the contact conditions are finished it is important to generate mesh again. After any change of boundary condition and geometry the mesh has to be generated again. It is done by the command **`Mesh | Generate mesh`** in the main menu or by pressing the key Ctrl and 'g' at the same time.

If the new mesh is generated, the analysis can be started again. It is done by using the command **`Calculate | Calculate`** or by the clicking on the icon . After selecting this command, the program will start to generate the input files for each step of the non-linear analysis and then the AtenaWin window will appear and analysis will be in progress (see Figure 147).

4.2 AtenaWin interface description

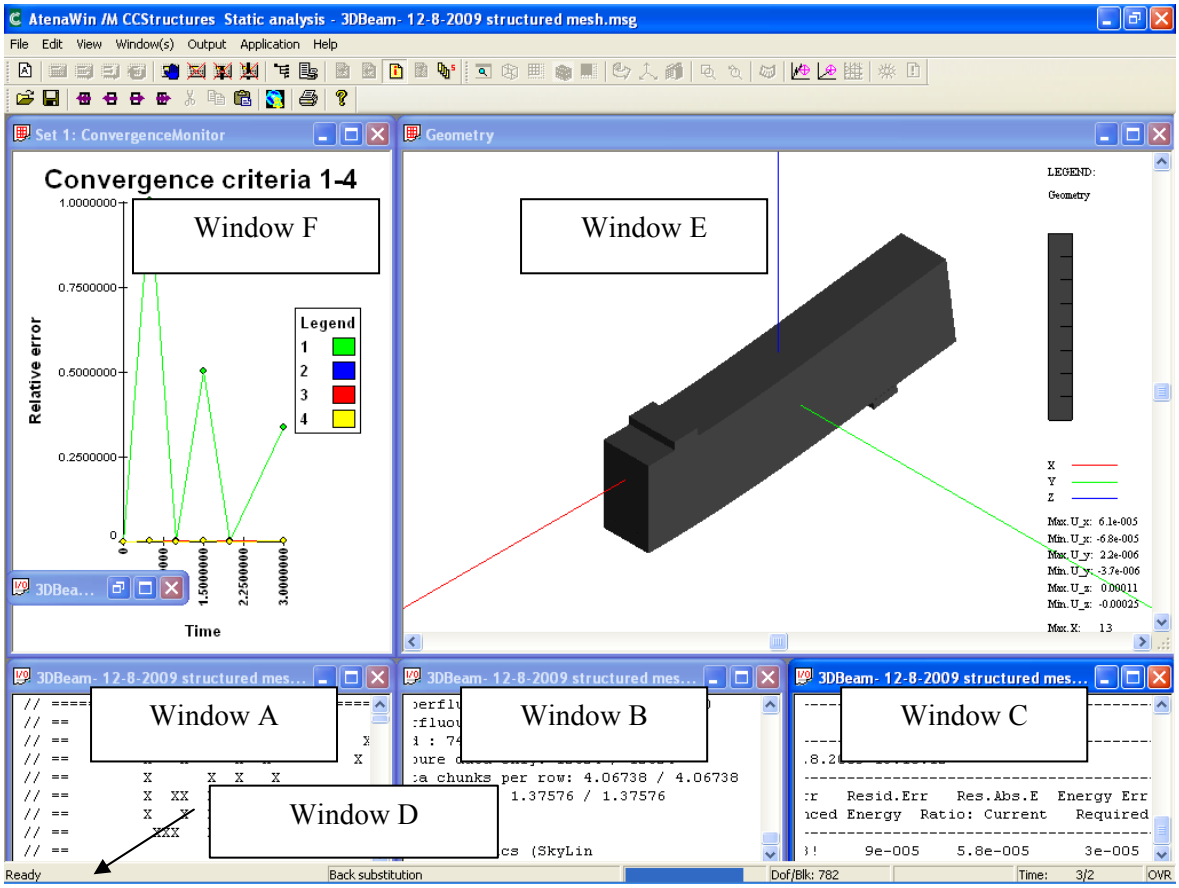


Figure 147: The analysis in progress

Basic description of the AtenaWin interface:

Window A: contains specification of input commands that describes a problem

Window B: contains results from the current analysis

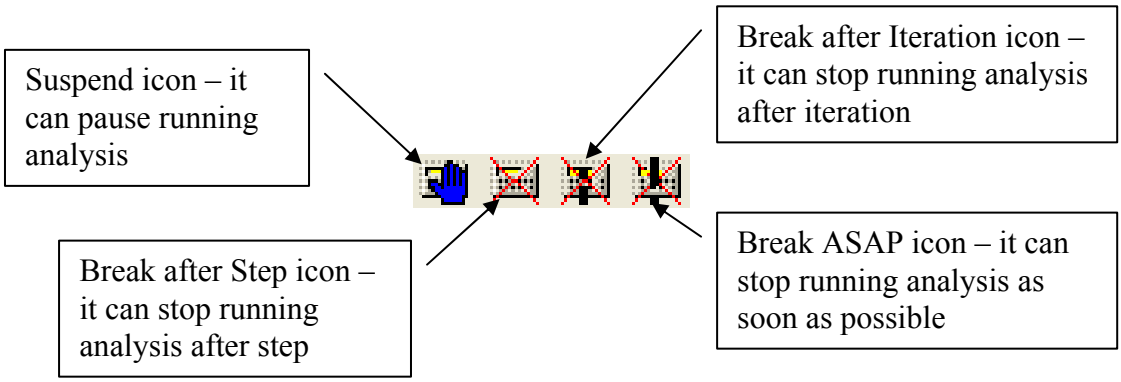
Window C: contains important messages from ATENA kernel sent during analysis

Window D: contains error and warning from ATENA kernel sent during analysis

Window E: contains graphical representation of the analysed structure

Window F: contains relative error – time graph

When the analysis is running it is possible to stop the calculation. For that can be used **`Application | Suspend/Break ASAP/Break after Iteration/Break after Step`** command in the main menu or icons of the Execution toolbar:


Suspend icon – it can pause running analysis
Break after Iteration icon – it can stop running analysis after iteration
Break after Step icon – it can stop running analysis after step
Break ASAP icon – it can stop running analysis as soon as possible

For detailed description of the AtenaWin user interface it is recommended to read AtenaWin Mannual [7].

4.3 Load-displacement graph

During the running analysis it is very useful to see the progress of the load and displacement. The progress of the load and displacement are monitor by defined monitors.

By the clicking on the icon of the Monitor set 2 the window for the graph will appear (Monitor set 2 is monitoring after step. Monitor set 1 is monitoring after iteration. In this case the monitors where set after steps therefore the Monitor set 2 should be selected).

The graph parameters are not defined yet therefore this window is empty. After executing of the graph window of the monitor set 2 the Graph series icon become to be active (see Figure 148). After clicking on that icon of the Graph series, the window for the definition of the graph parameters will appear. By the clicking on the button **`New Series`**, the parameters of the graph can be defined (see Figure 149).

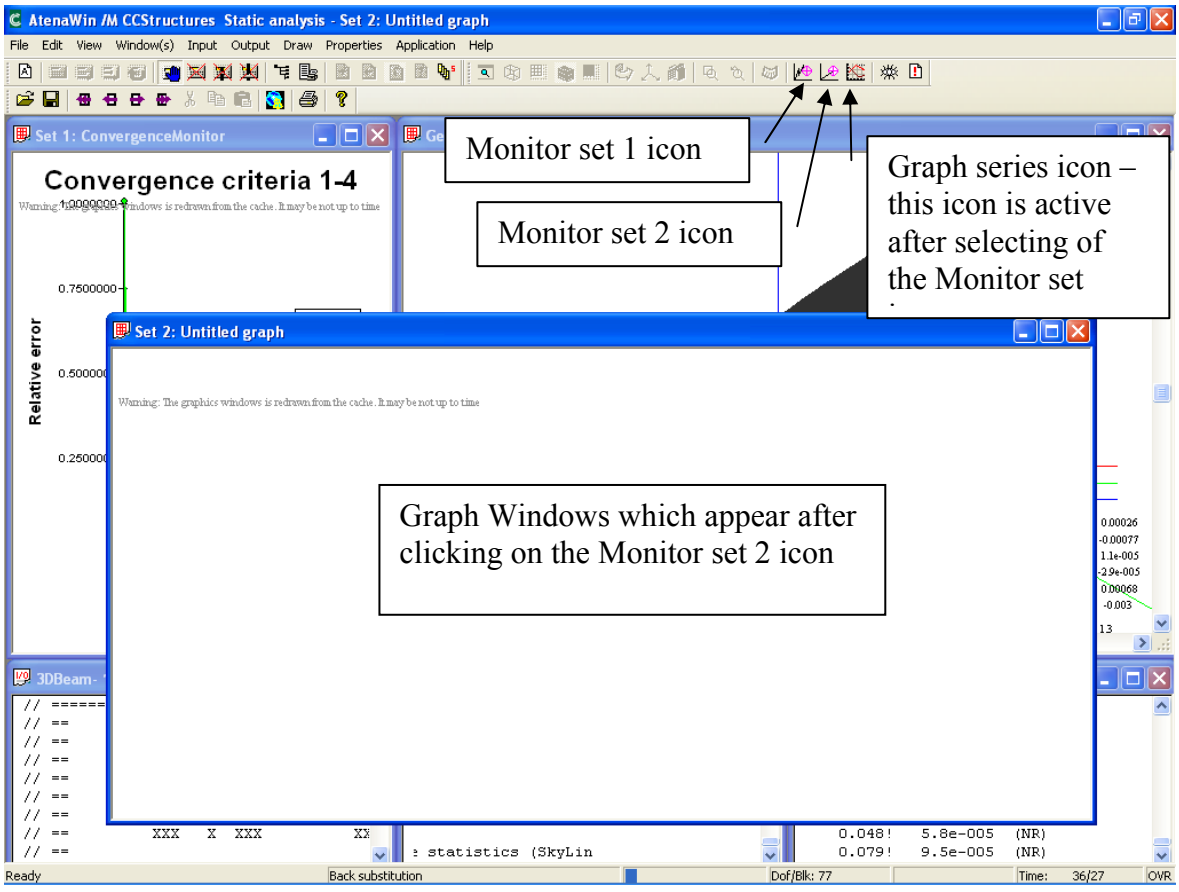

AtenaWin /M CCStructures Static analysis - Set 2: Untitled graph
File Edit View Window(s) Input Output Draw Properties Application Help
Set 1: ConvergenceMonitor
Convergence criteria 1-4
Relative error
Monitor set 1 icon
Monitor set 2 icon
Graph series icon – this icon is active after selecting of the Monitor set
Set 2: Untitled graph
Warning: The graphics windows is redrawn from the cache. It may be not up to time
Graph Windows which appear after clicking on the Monitor set 2 icon
Ready
Back substitution
Dof/Blk: 77
Time: 36/27
OVR

Figure 148: The execution of the graph

The name of the graph should be **L-D**. As name of the horizontal value should be selected **Deflection_DISPLACEMENT** and factor should be **-1**. Name of the vertical value should be selected **Load_REACTIONS** and factor will also **-1**. All definition of the graph parameters is finished by clicking on the **OK** button. After this the L-D graph is shown on the top of the AtenaWin interface. This graph is showing actual stage of the running analysis and it changes according the running analysis.

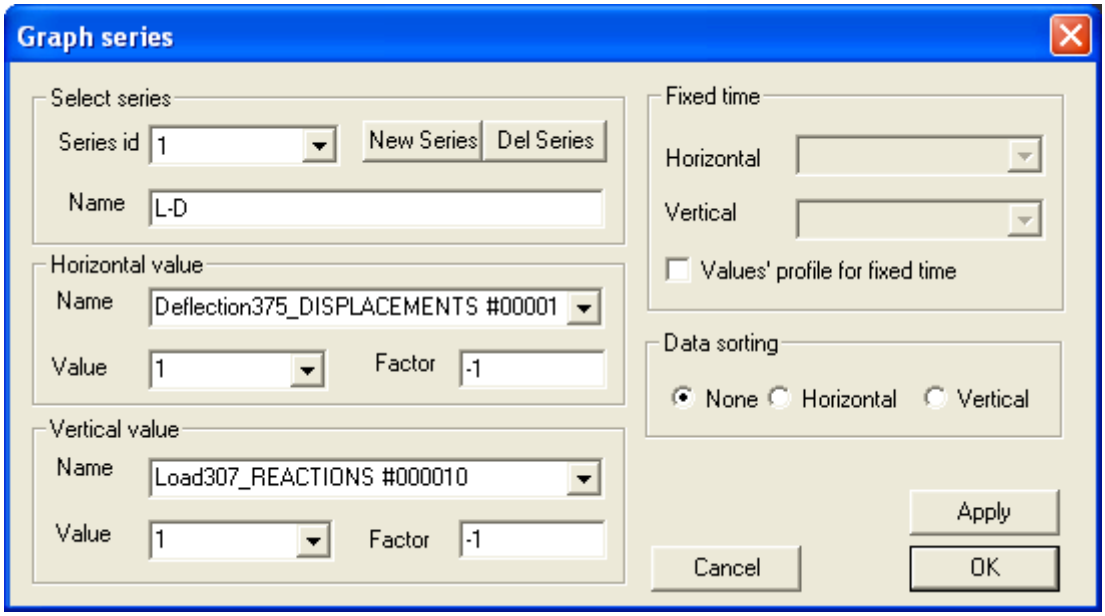


Figure 149: The graph series definition

Parameter input:
Name: L-D
Horizontal value
Name: Deflection_DISPLACEMENT
Factor: -1
Vertical value
Name: Load_REACTIONS
Factor: -1

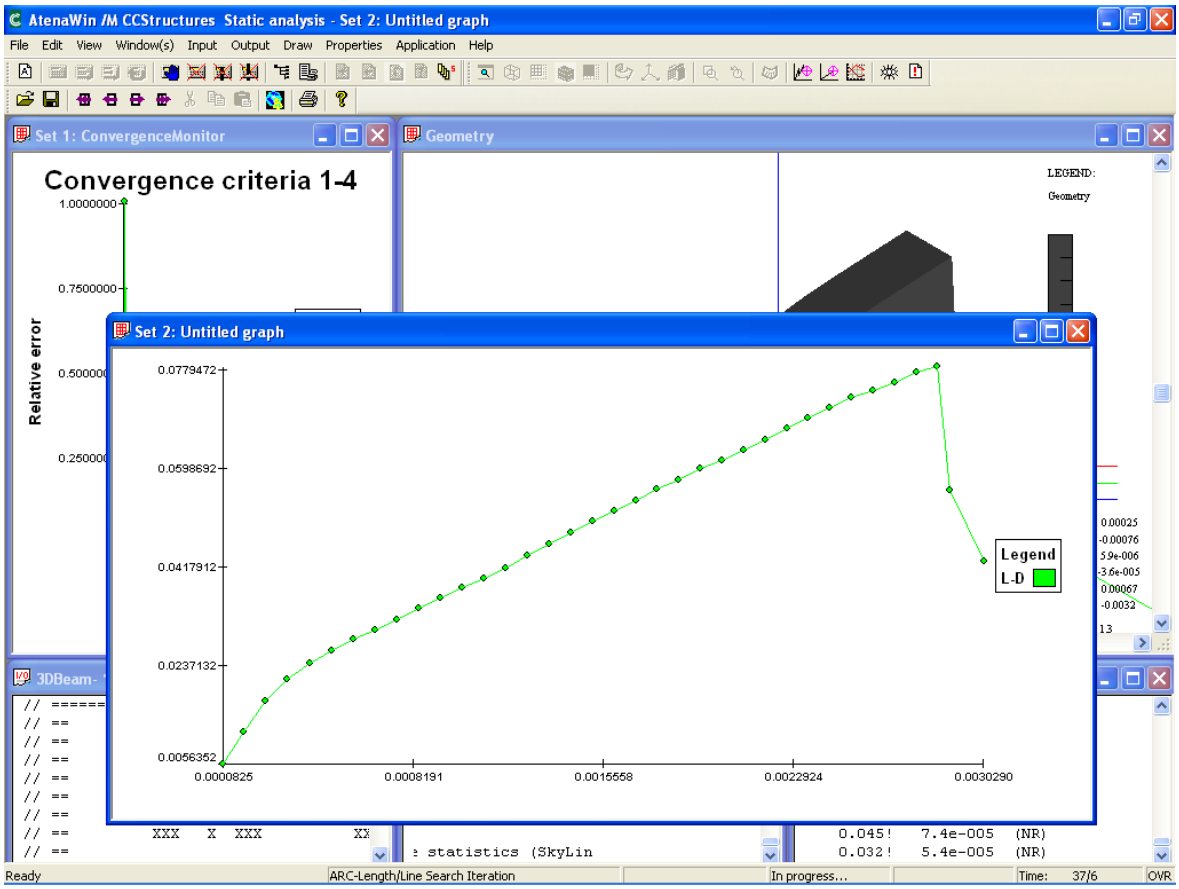


Figure 150: The L-D diagram showing stage of the running analysis

The graph parameters were defined. Now, the graph properties should be set. It is done by the selecting of the Draw properties icon . After that the graph property window will appear and properties can be described. The name of the graph should be **`L-D`**, name of the horizontal value should be **`Displacement`** and name of the vertical value should be **`Load`**. It can be useful to change scale of axis (in this case it has been chosen to change scale on horizontal value to **`10`** and on vertical axis to **`6`**). The graph properties are described in the Figure 151 and the definition is finished by the clicking on the **`Apply`** button (see Figure 151).

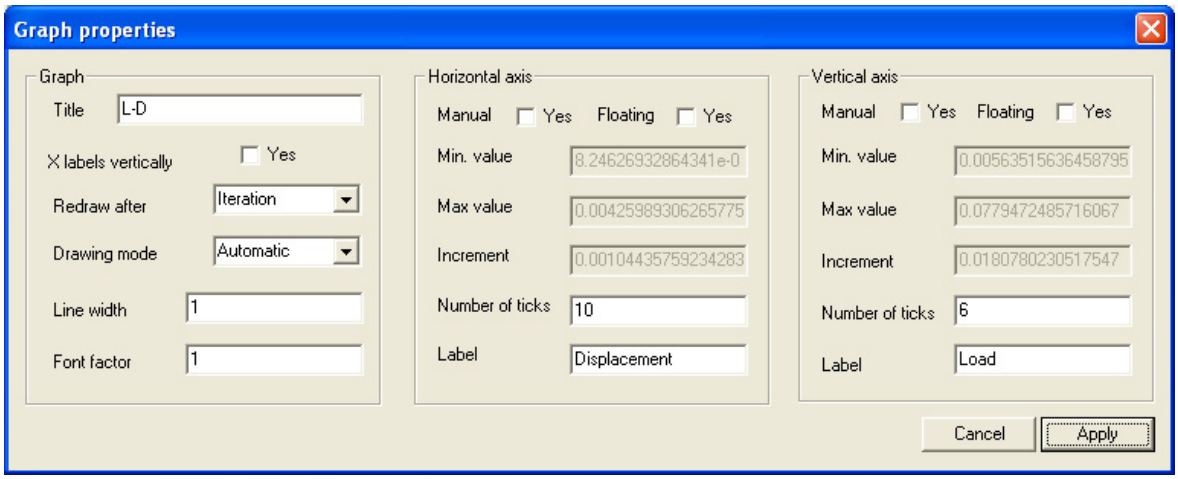


Figure 151: The graph properties definition

Parameter input:

Title:	L-D
Horizontal axis	
Number of ticks:	10
Label:	Displacement
Vertical value	
Number of ticks:	6
Label:	Load

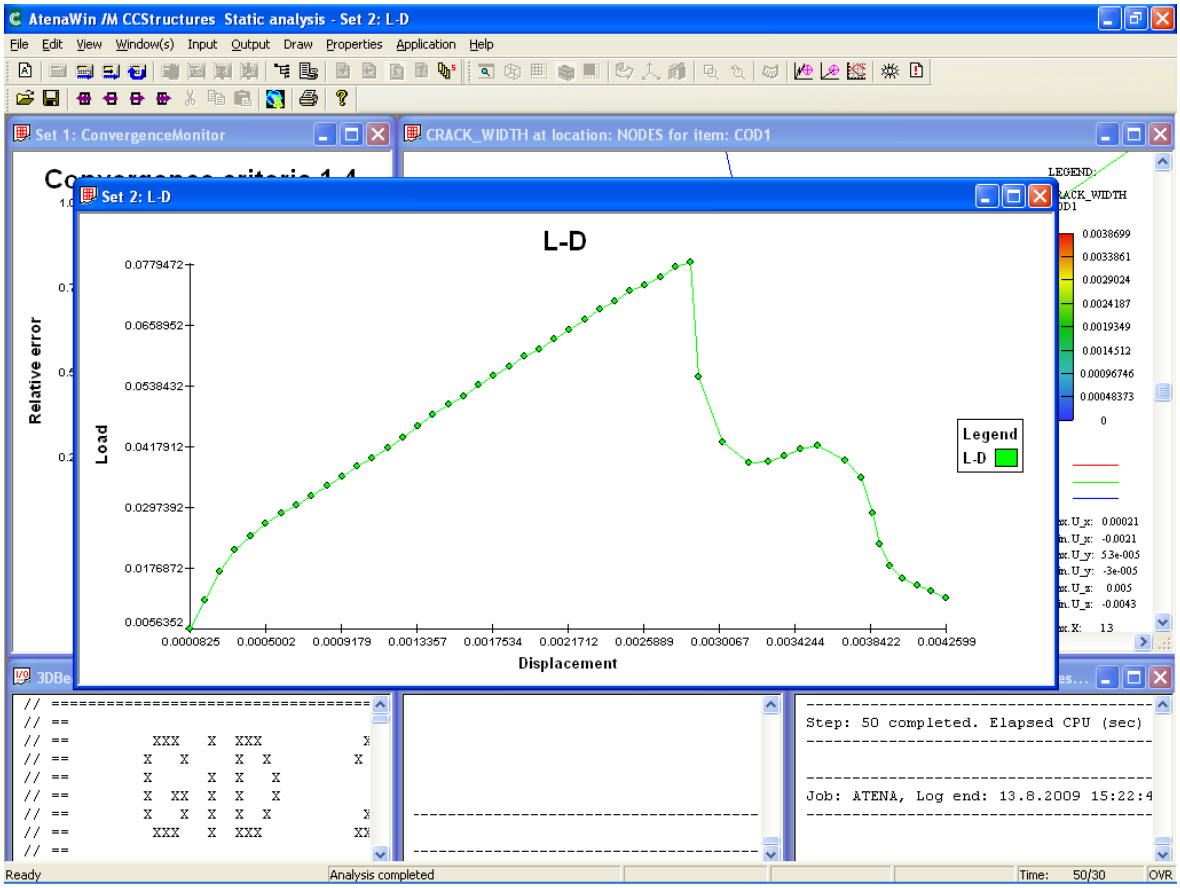


Figure 152: The defined L-D graph

This graph can be saved by the command **`File | Save Figure Settings`** in the main menu.

Detailed description of the L-D graph creation can be found in the AtenaWin Manual [7] chapter 5.

When there are more windows in the layout of the AtenaWin the command Default layout 1 and Default Layout 2 in the main menu can be used to organize all windows. After selecting the option Default layout 2 (**`Window(s) | Default layout 2`** in the main menu), all widows of the same type will be behind each other (see Figure 153). It means that all text windows are on the bottom of the basic layout, all geometry windows are on the right side and all graphs are on the left side. Individual windows can be switched by icons and .

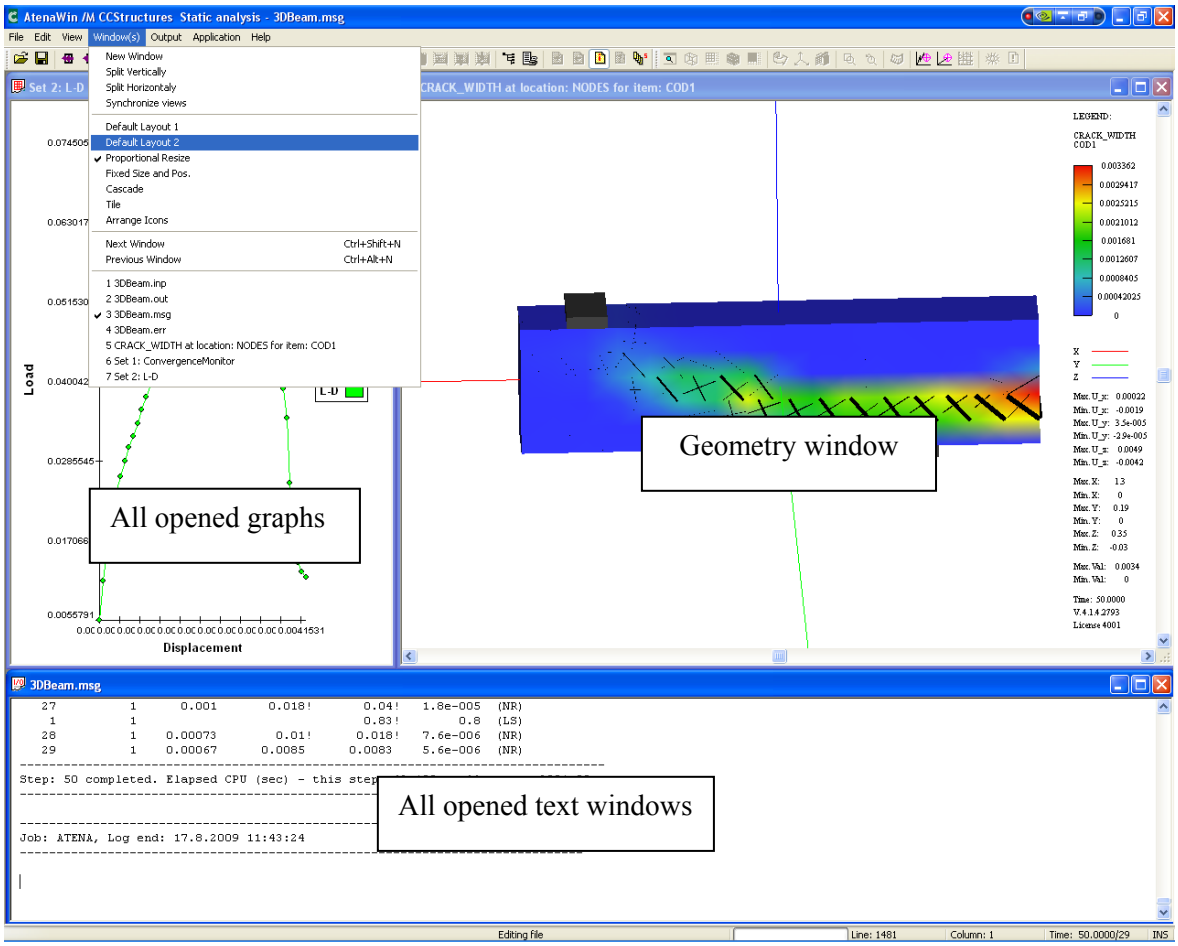


Figure 153: The executing of the Default layout 2

4.4 Crack width display

During the running analysis can be also good to display crack width in the Geometry window. When this window is active the all icons of Graphics Toolbar are active too. The executing of displaying of the crack width is done by clicking on the Post-processor data icon (see Figure 154). After that the Post-processor data window will appear and the width crack display can be defined (see Figure 155).

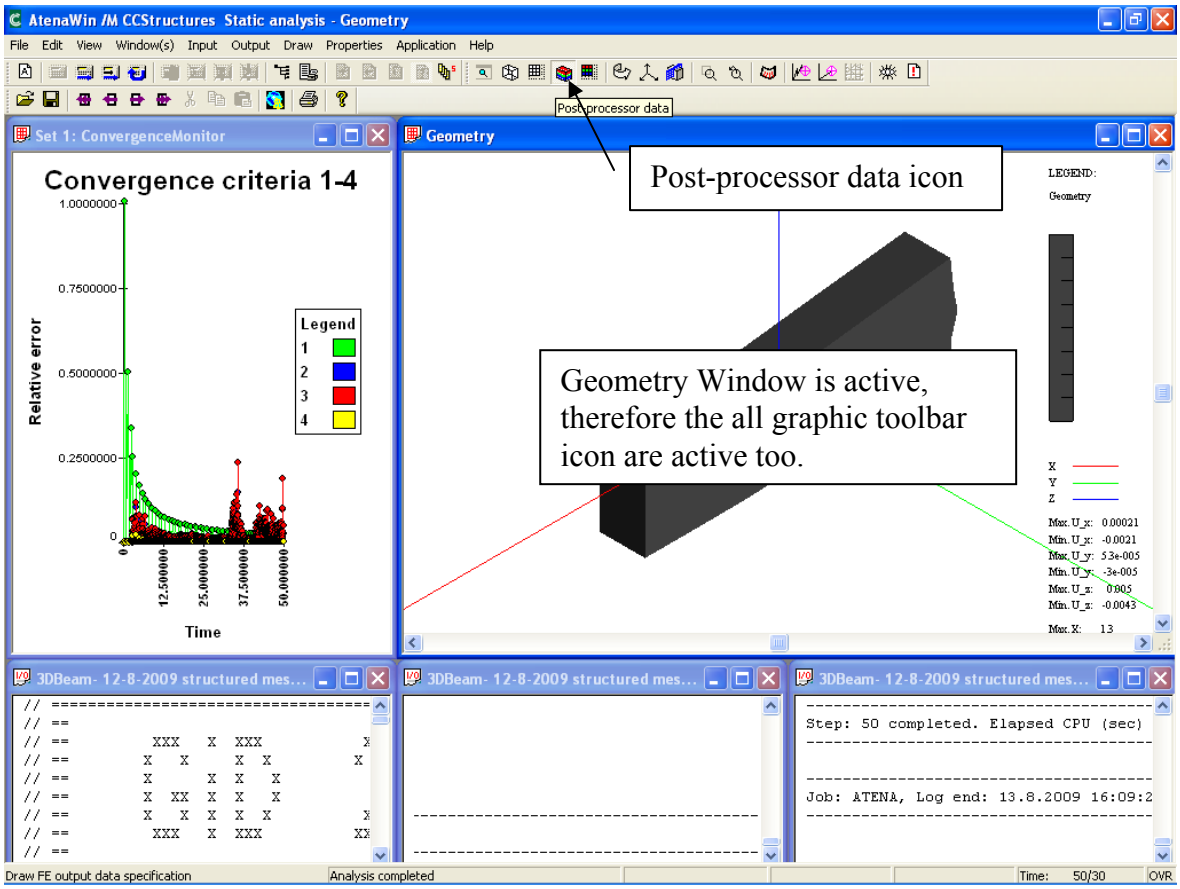


Figure 154: The active geometry window

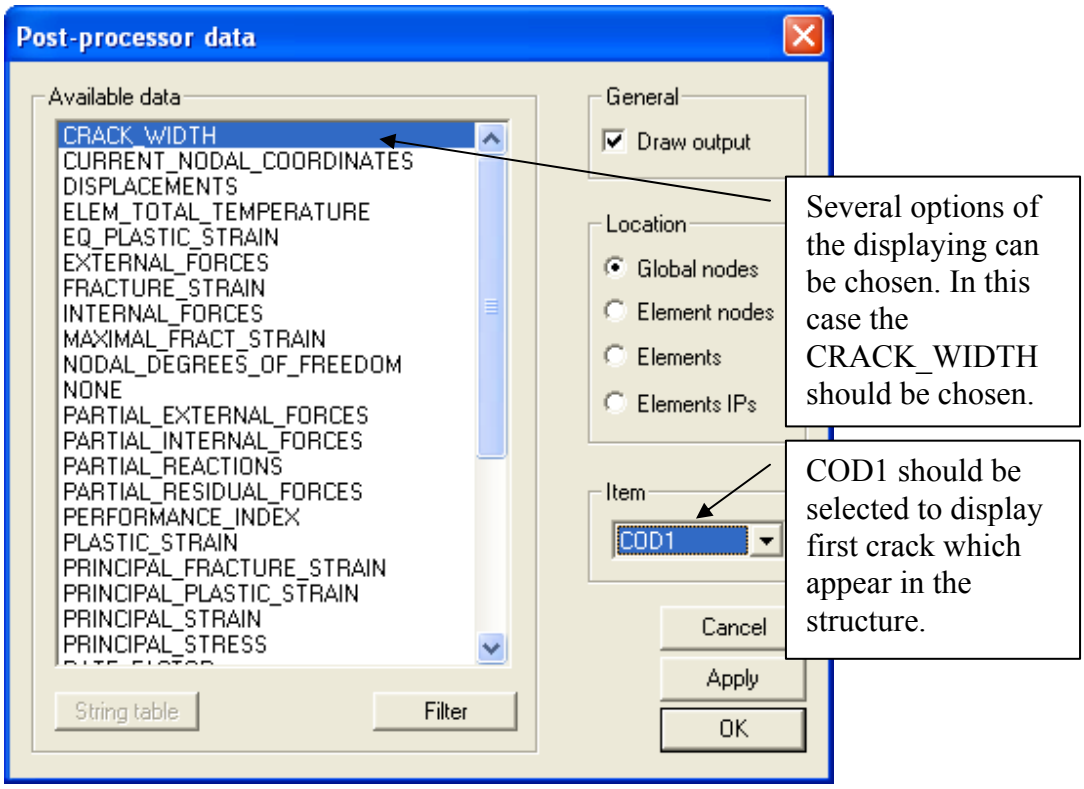


Figure 155: The crack width display

The button **OK** has to be selected to complete the displaying of the cracks width (see Figure 156).

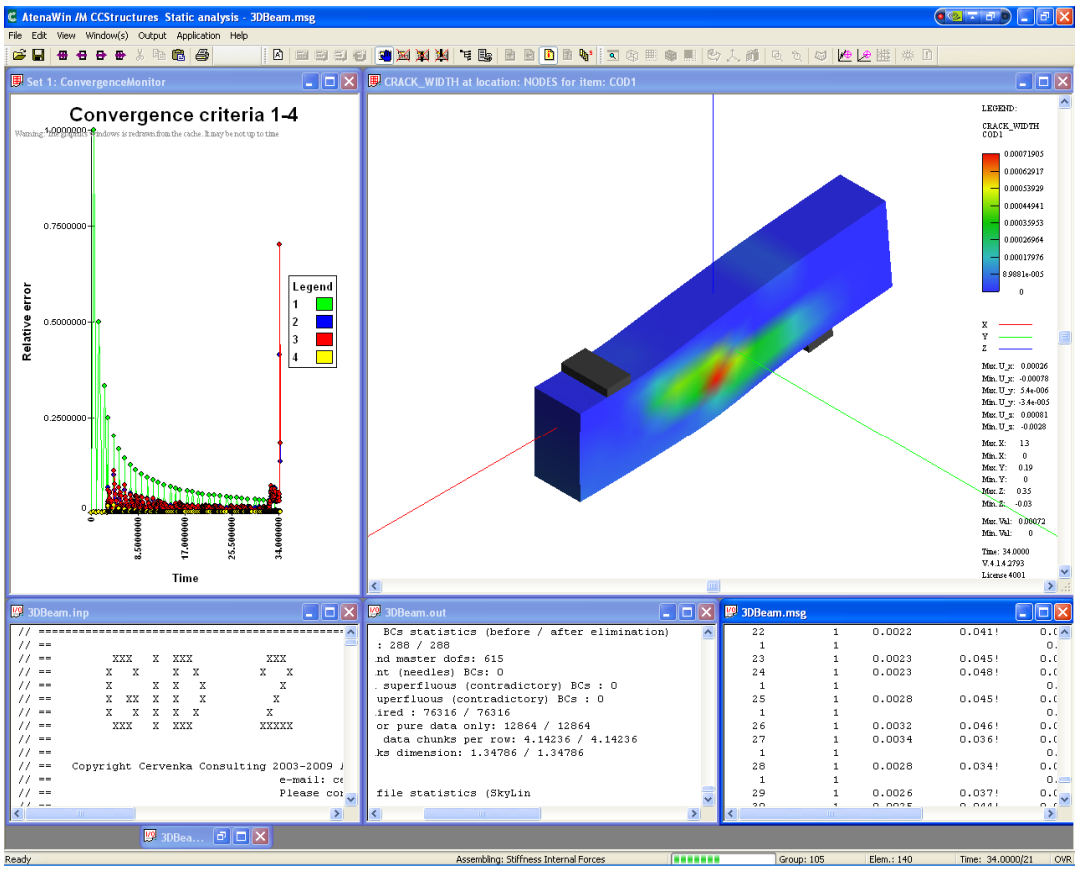


Figure 156: The crack width display in the geometry window

For better view the model can be rotated. It is done by the clicking on the Model transformation icon and then by the setting of the degree of the rotation in the each axis direction (see Figure 157).

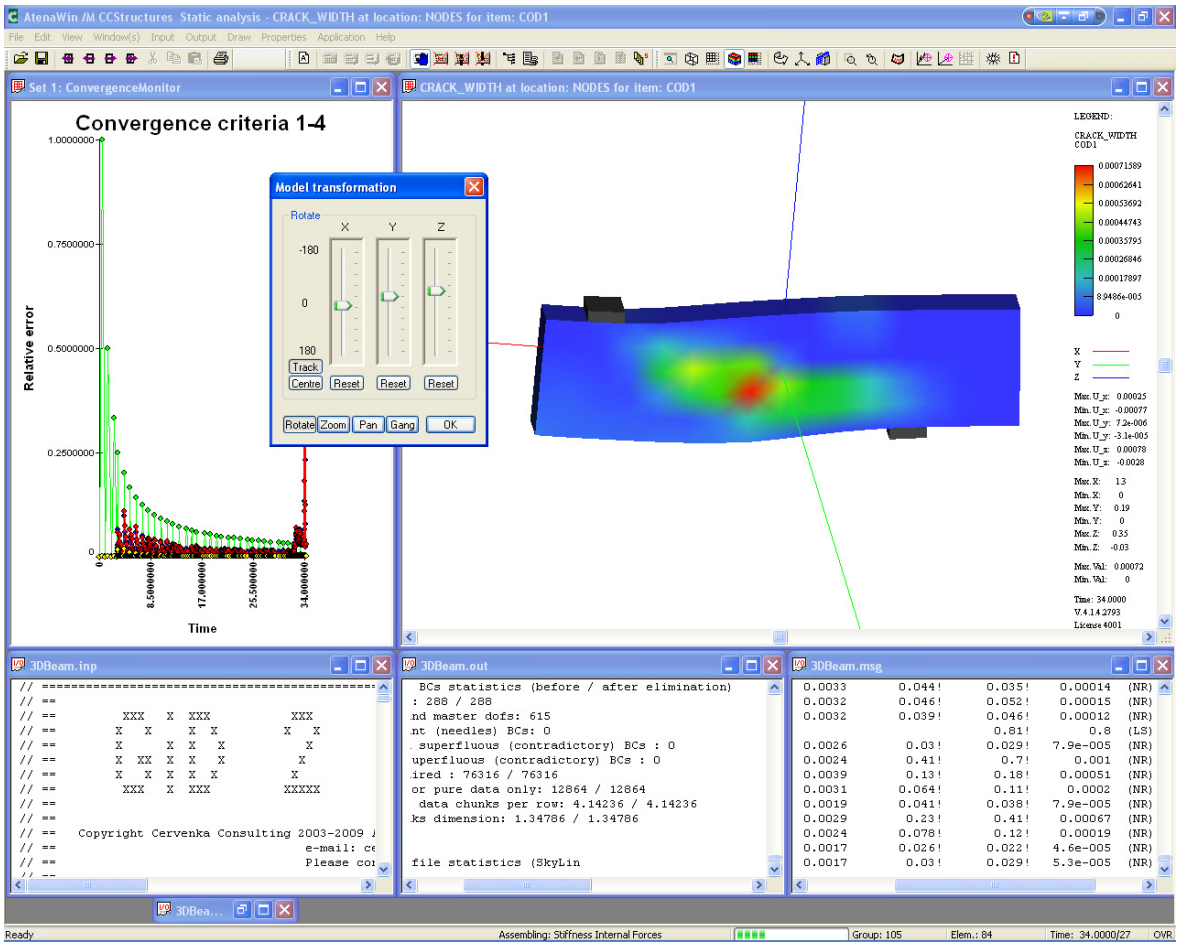


Figure 157: The crack width display and the rotation of the model

Also for better view the model can be displayed undeformed. It is done by the clicking on the Draw properties icon and then by the setting of the Draw undeformed checkbox (see Figure 158). This command is completed by selecting of the **Apply** button (see Figure 159 and Figure 160).

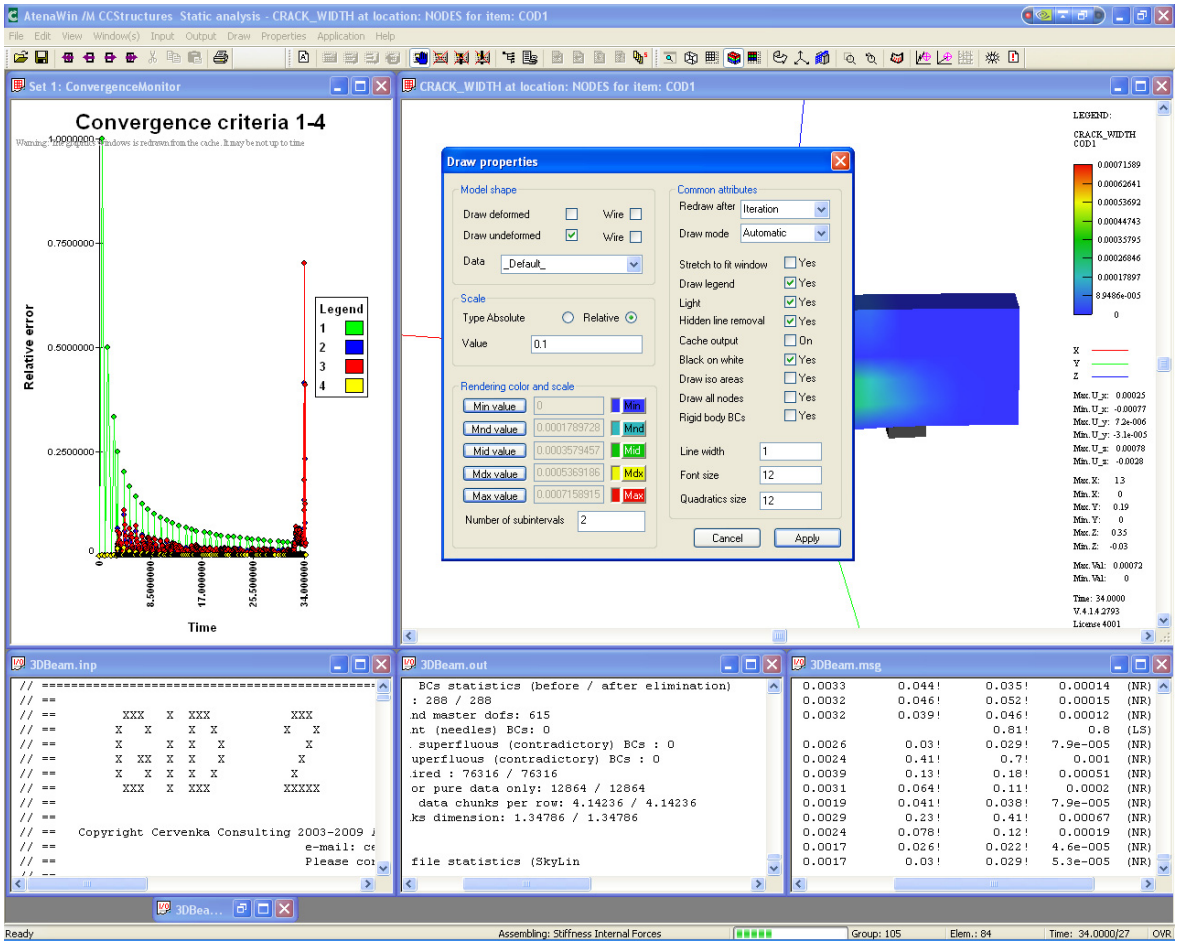


Figure 158: The setting of the display of the undeformed model

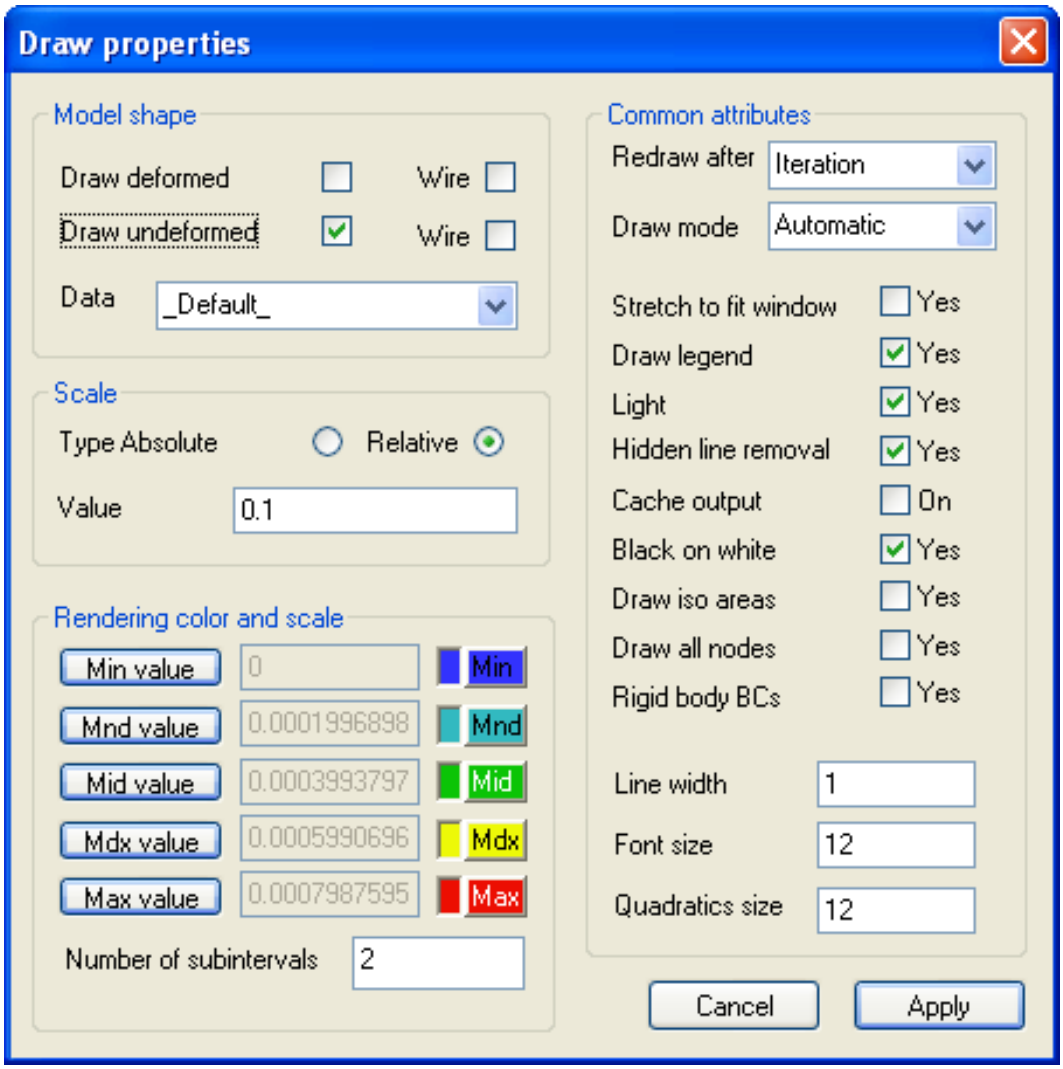


Figure 159: The setting of the display of the undeformed model

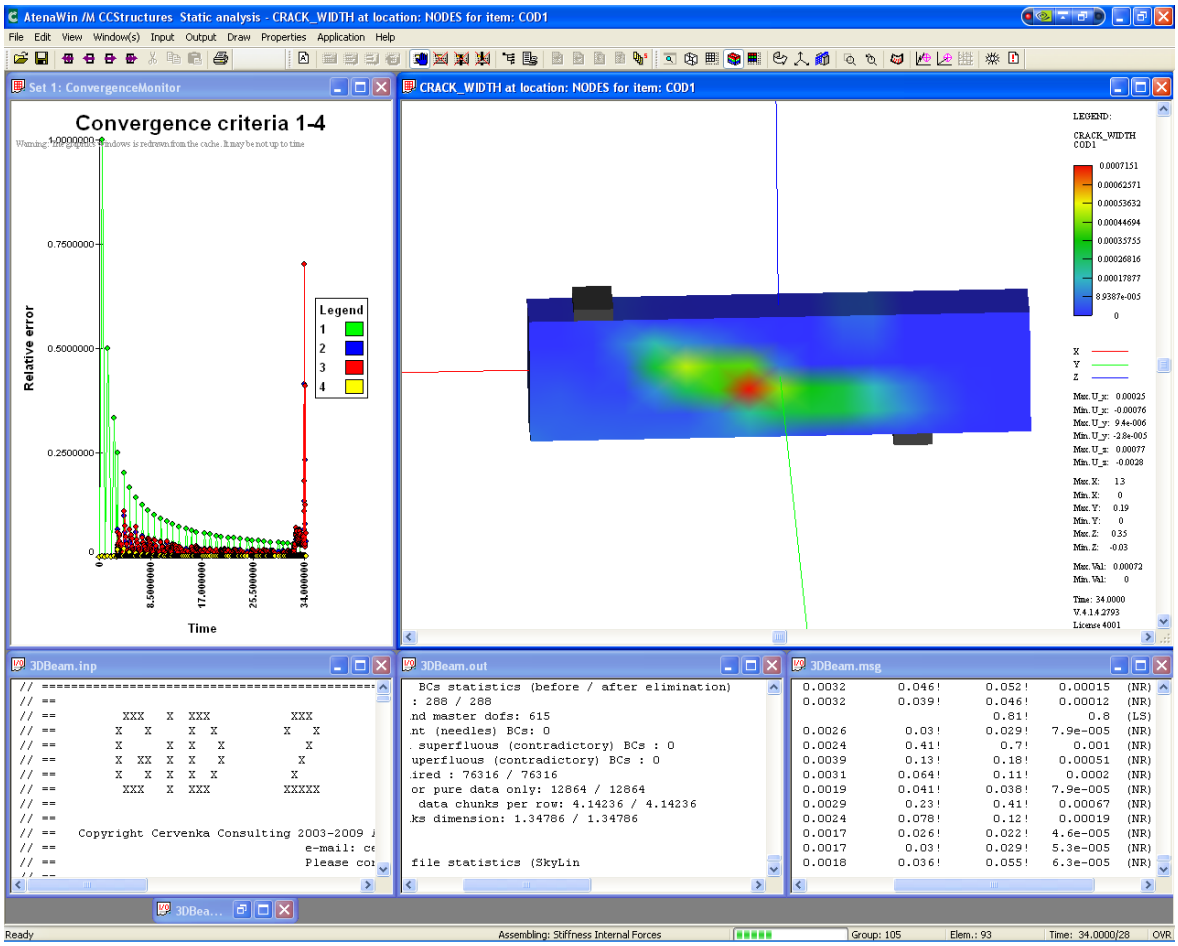


Figure 160: The display of the undeformed model

Another option for the better display the crack width can be drawn. It is done by the clicking on the Draw Crack Option icon (see Figure 161) and then by the setting of the crack width and multiplier (see Figure 162). This command is completed by selecting of the **`Apply`** button (see Figure 163).

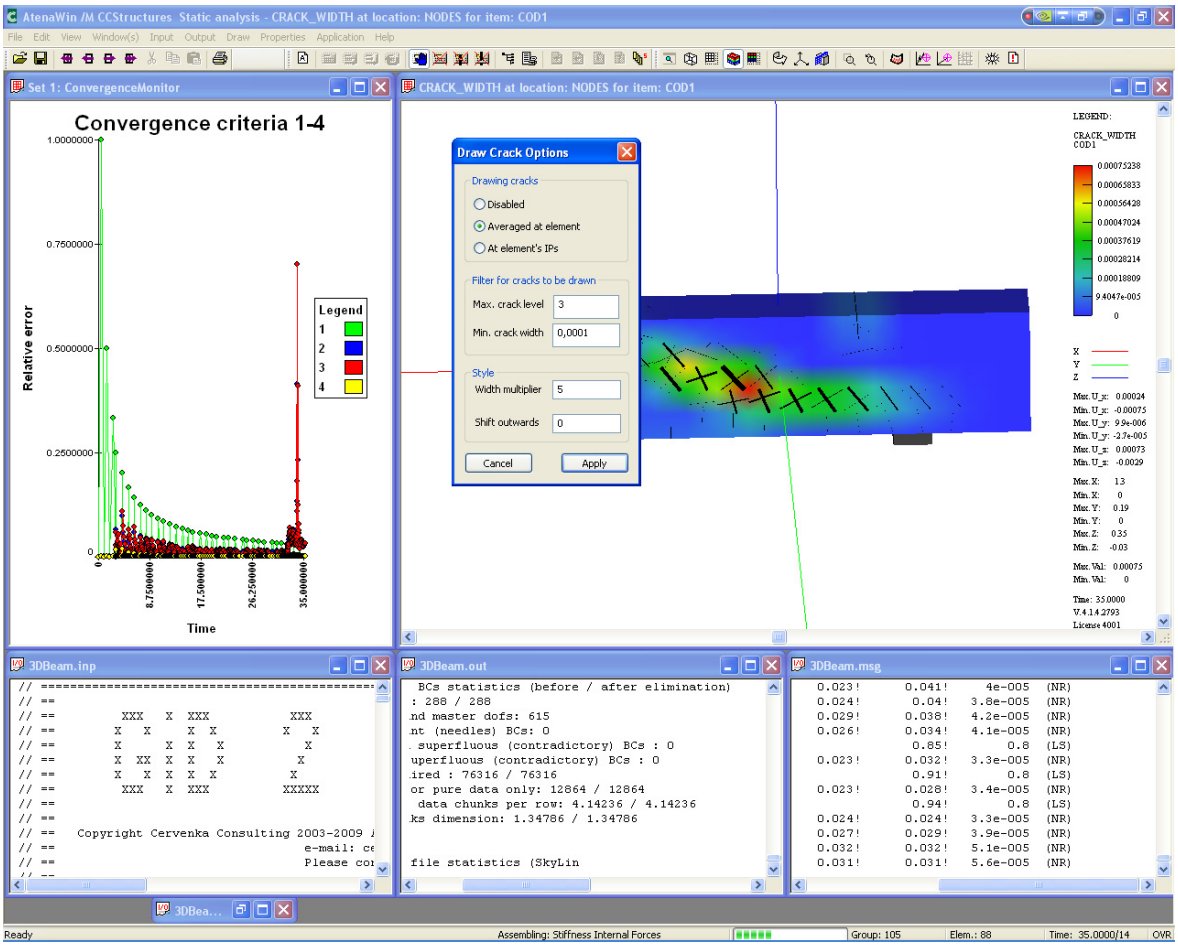
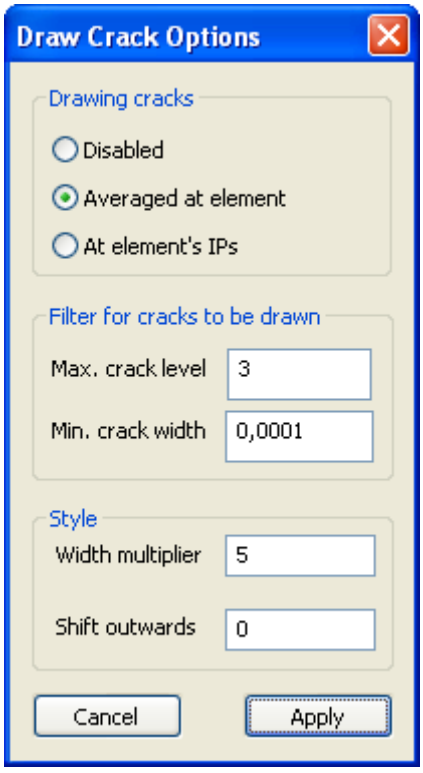


Figure 161: The executing of the Draw Crack Option



Parameter input:

Averaged at element

Max. crack level:	3
Min. crack width:	0,0001
Width multiplier:	5

Figure 162: The setting of the Draw crack option

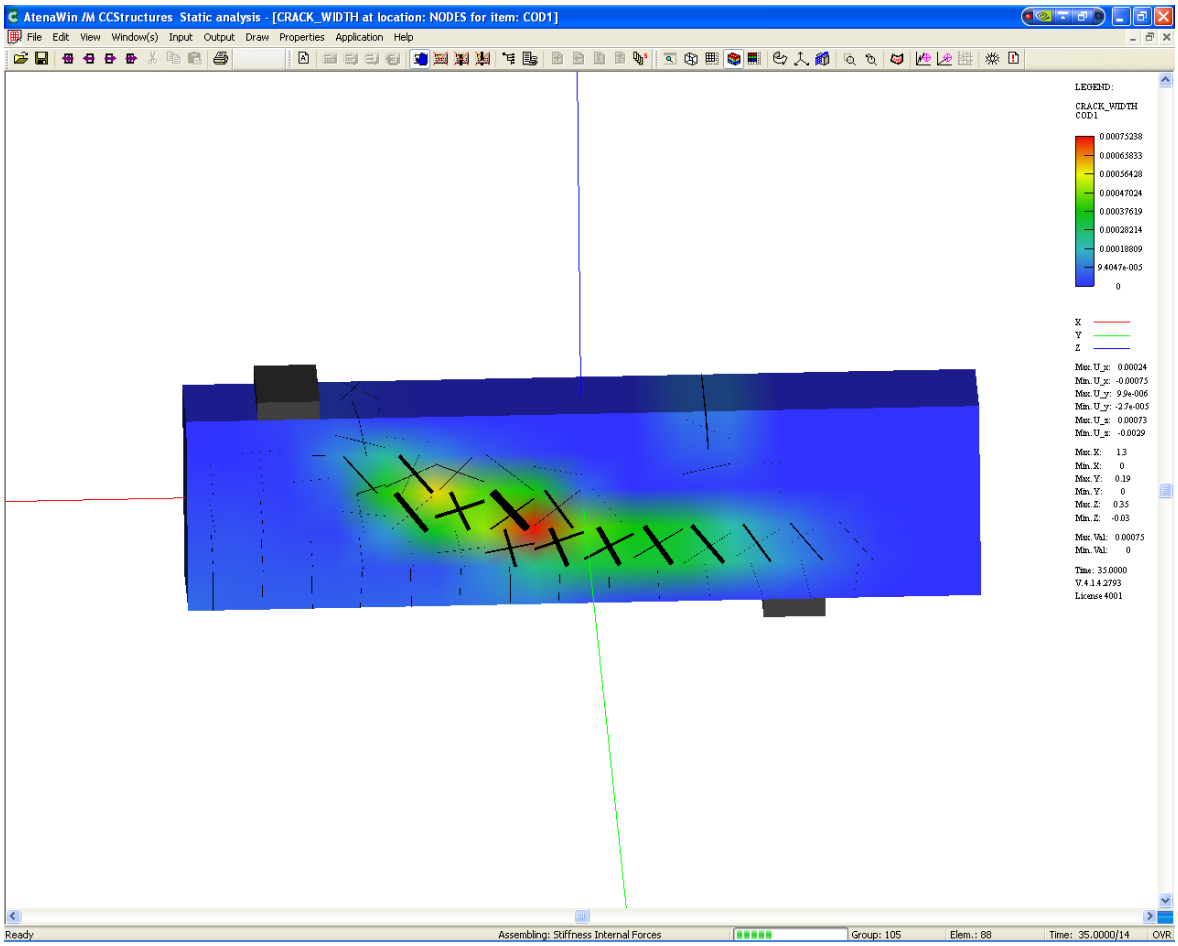


Figure 163: The drawn crack width

All of these options of the crack width display, display and draw are in the fact the post processing features, but they can be used during the running of the analysis. During running analysis the all **AtenaWin** post-processing capabilities can be used. For more information it is recommended to study AtenaWin Manual [7].

5. POST-PROCESSING

The created model can be post-process in the **AtenaWin** or in the **GiD**.

5.1 GiD post-processing

After finished analysis AtenaWin window can be closed. The program asks if all changes should be saved. Then button **`Yes`** should be selected in all cases.

Then back in the GiD interface the process info will appear. Through this dialog the program asks if the process of the analysed problem is finished or if the post-process should be started. The button **`Postprocess`** should be selected (see Figure 164).

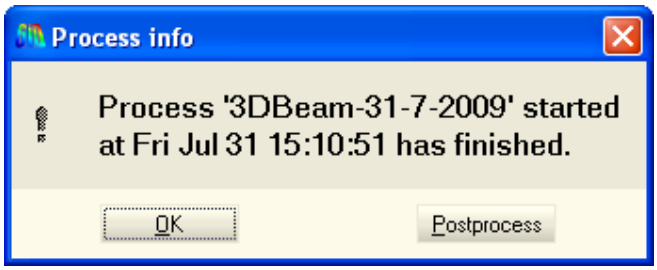


Figure 164: The button Postprocess should be pressed

But before any postprocessing features can be started the results from the **AtenaWin** has to be imported into **GiD**.

It is done by the clicking on the Import results from **AtenaWin** icon . Then the process of importing will start (see Figure 166) and when it is finished the model changes its colours (see Figure 167).

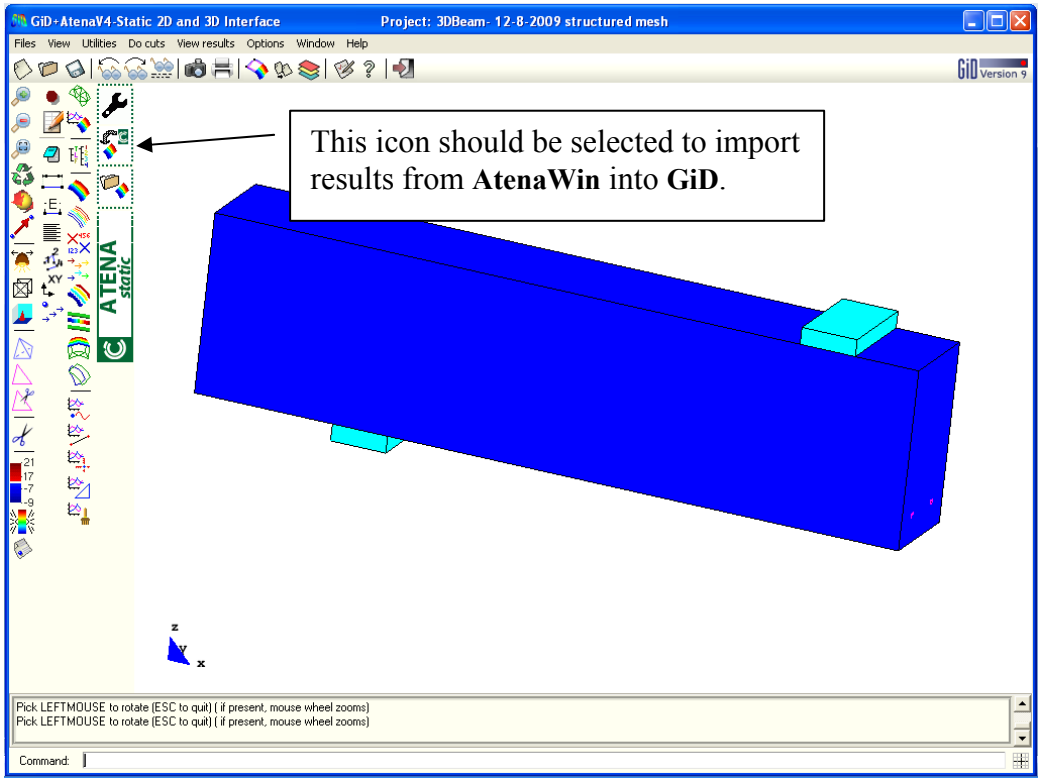

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- 12-8-2009 structured mesh
Files View Utilities Do cuts View results Options Window Help
GiD Version 9
This icon should be selected to import results from AtenaWin into GiD.
ATENA static
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Pick LEFTMOUSE to rotate (ESC to quit) (if present, mouse wheel zooms)
Command:

Figure 165: The GiD postprocessor interface

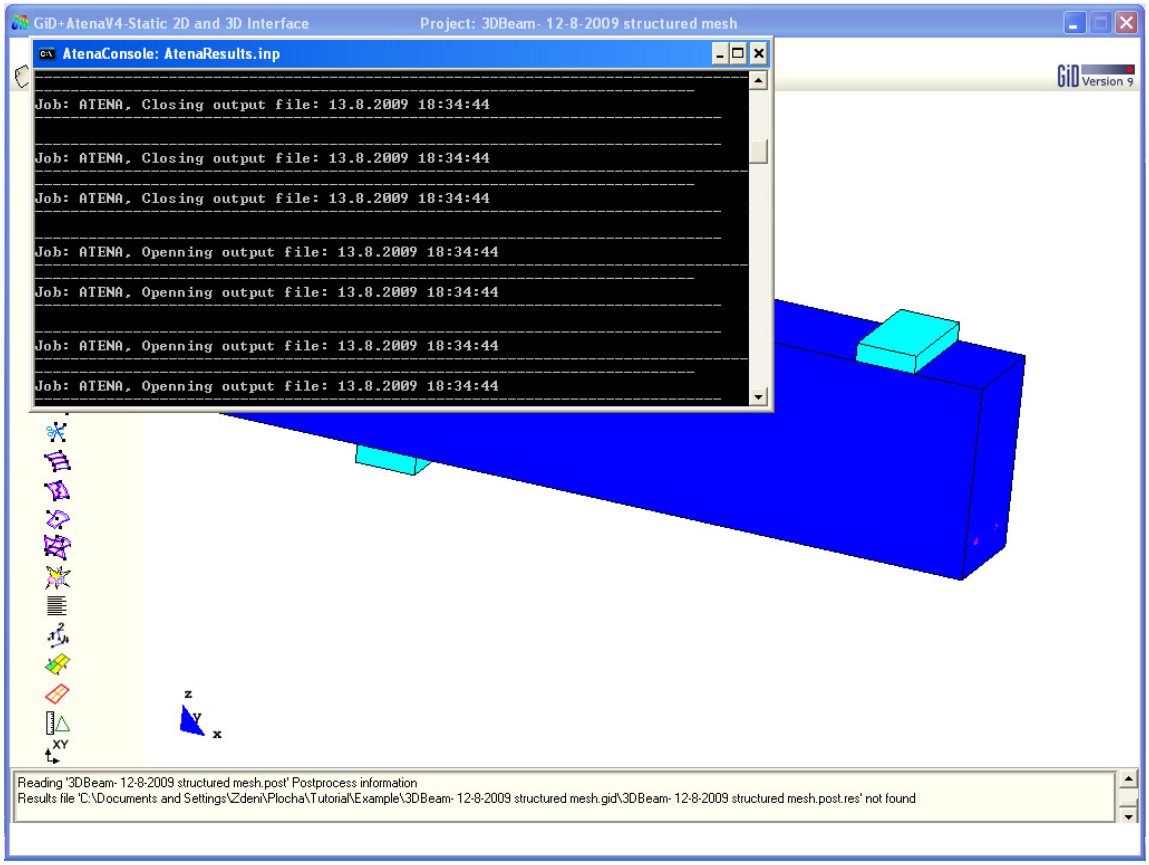

GiD+AtenaV4-Static 2D and 3D Interface
Project: 3DBeam- 12-8-2009 structured mesh
AtenaConsole: AtenaResults.inp
Job: ATENA, Closing output file: 13.8.2009 18:34:44
Job: ATENA, Closing output file: 13.8.2009 18:34:44
Job: ATENA, Closing output file: 13.8.2009 18:34:44
Job: ATENA, Openning output file: 13.8.2009 18:34:44
Job: ATENA, Openning output file: 13.8.2009 18:34:44
Job: ATENA, Openning output file: 13.8.2009 18:34:44
Job: ATENA, Openning output file: 13.8.2009 18:34:44
GiD Version 9
Reading '3DBeam- 12-8-2009 structured mesh.post' Postprocess information
Results file 'C:\Documents and Settings\Zdeni\Plocha\Tutorial\Example\3DBeam- 12-8-2009 structured mesh.gid\3DBeam- 12-8-2009 structured mesh.post.res' not found

Figure 166: The importing of the results from AtenaWin into GiD

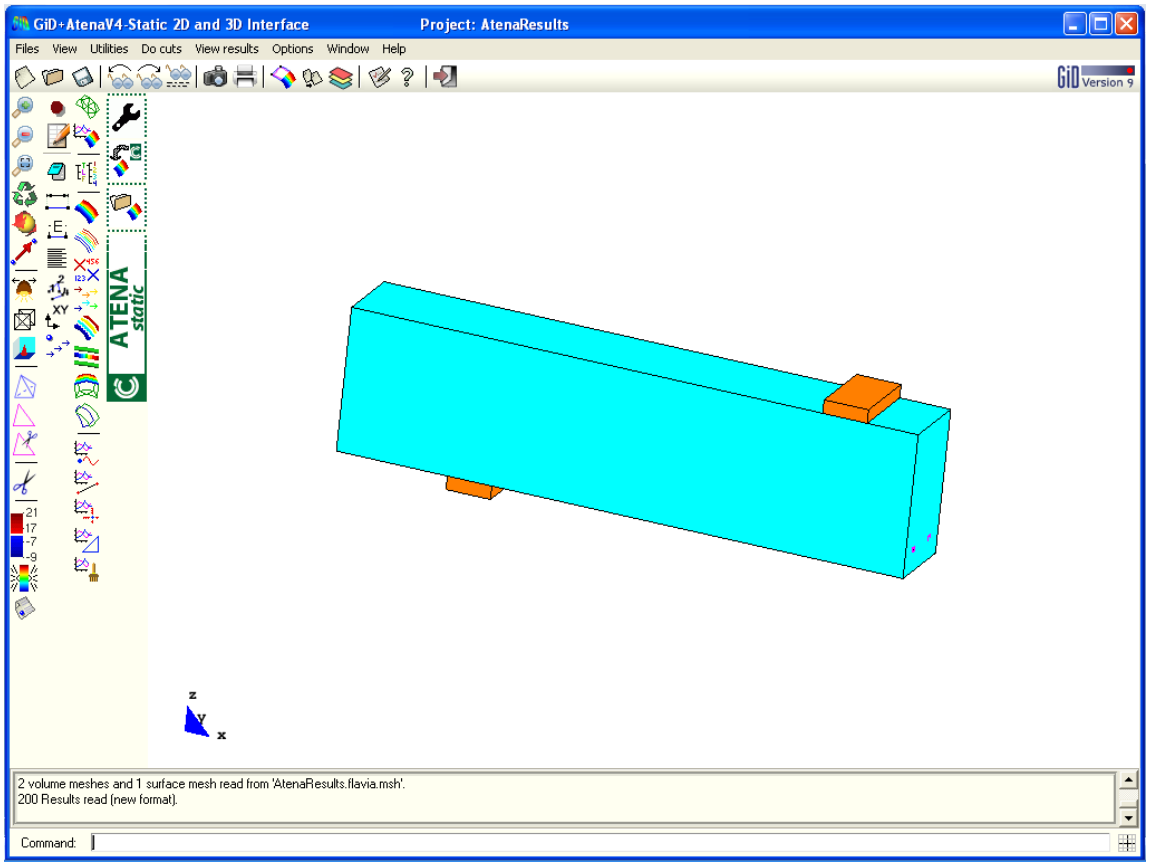

GiD+AtenaV4-Static 2D and 3D Interface
Project: AtenaResults
Files View Utilities Do cuts View results Options Window Help
GiD Version 9
ATENA static
2 volume meshes and 1 surface mesh read from 'AtenaResults.flavia.msh'.
200 Results read (new format).
Command:

Figure 167: The importing of the results from AtenaWin were finished

After importing data from **AtenaWin**, the post-processing can be started. Let's display cracks like in previous chapter 4.4 of FE non-linear analysis in **AtenaWin**.

First of all it should be checked which step will be post-processed. It is done by selecting **`View Results | Default Analysis/Step | AtenaResults2GiD`** in the main menu or by the Default Analysis/Step icon . From the L-D graph is possible to see that structure failed after 30th step, therefore it is good to post-process for example 35th step (see Figure 168).

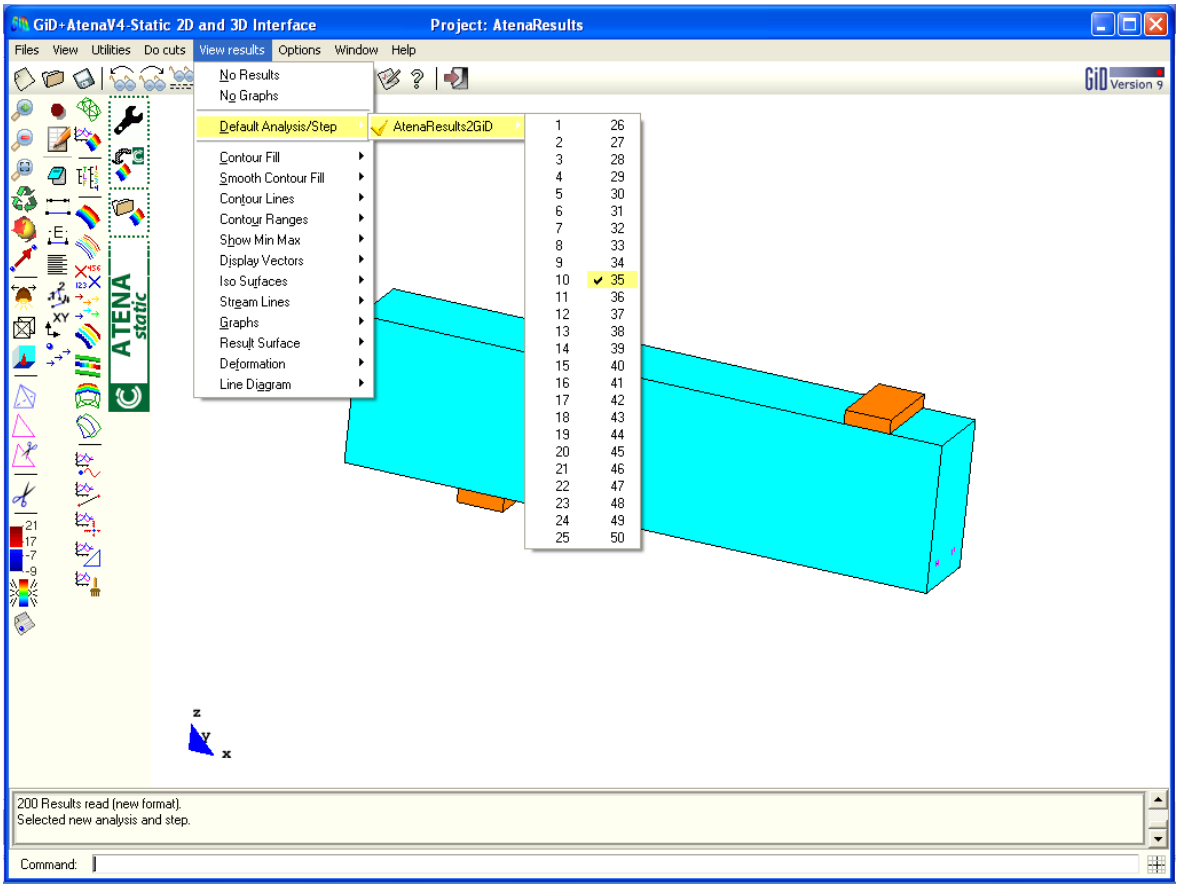


Figure 168: The selection of the step which should be post-processed

By the clicking on the Contour fill icon or by the selecting the command from main menu **`View results | Contour Fill | CRACK WIDTH | COD1`** crack width can be displayed like in previous chapter (see Figure 169).

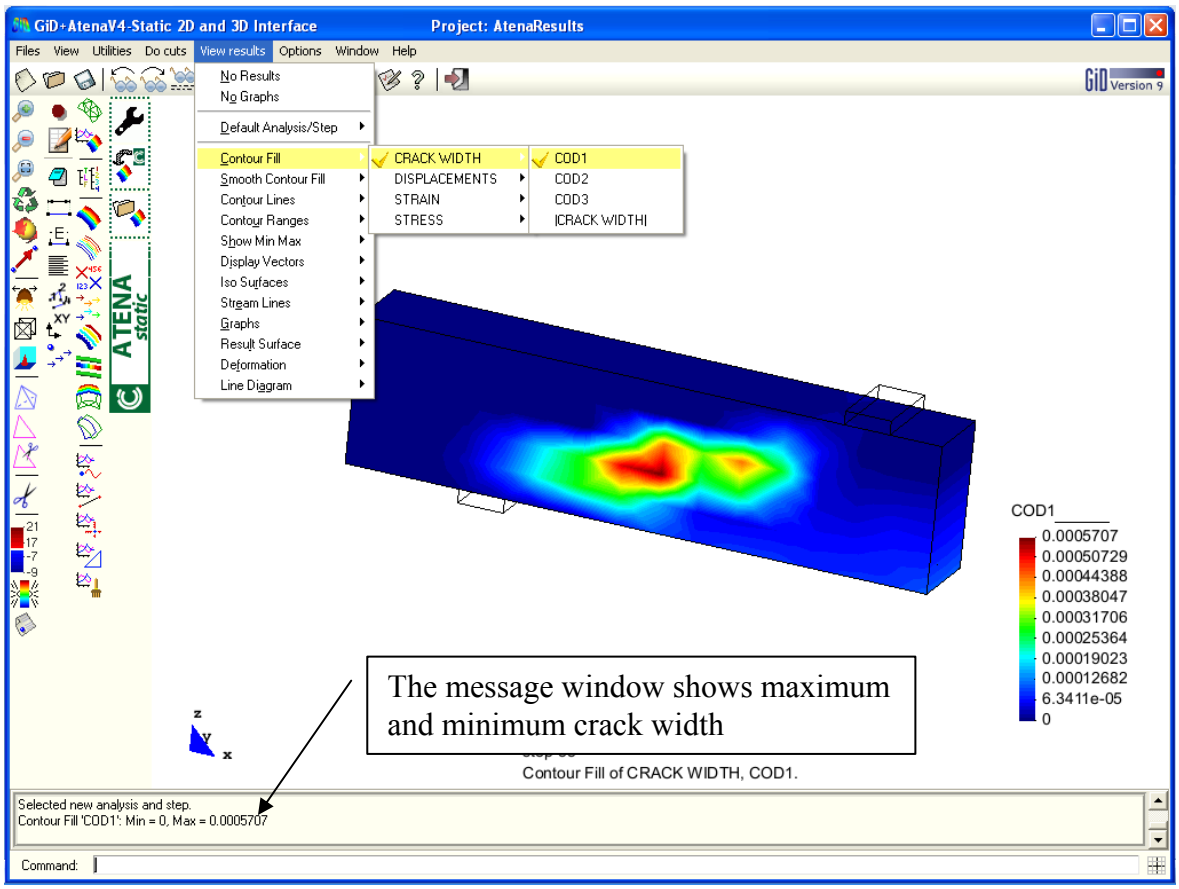


Figure 169: The display of the crack width

In the command for example Countour Fill the pull down menu offers options which can be displayed. To be able to post-process more results in **GiD**, the result quantities must be made available by selecting them in the pre-processing. Therefore first the program has to be switched to pre-processing.

It is done by selecting Toggle between pre and postprocess icon (see Figure 170). After the dialog window appear and the button **OK** should be pressed. The program switches into pre-processing. Then the command **`Data | Problem Data | Post Data`** can be selected in the main menu and the window for the definition of the post data will appear (see Figure 171).

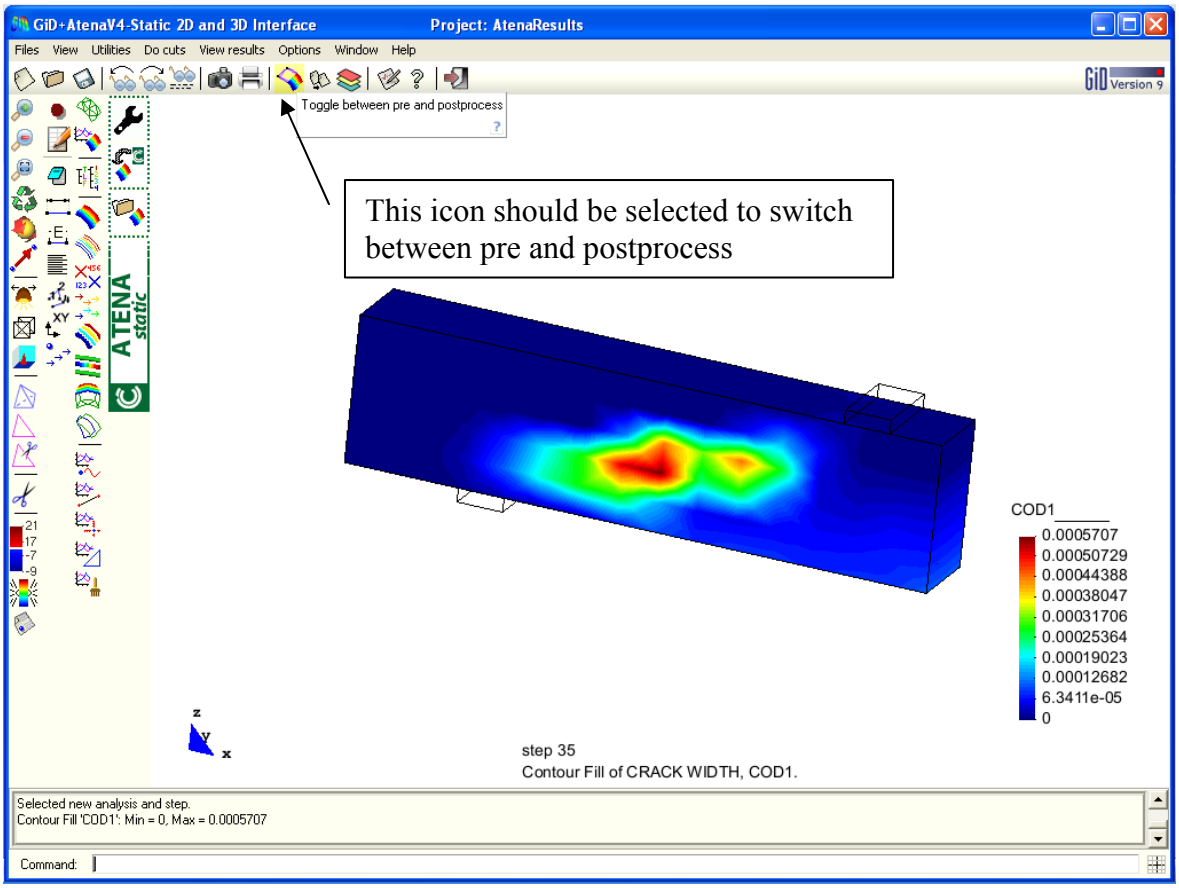


Figure 170: The Toggle between pre and postprocess icon

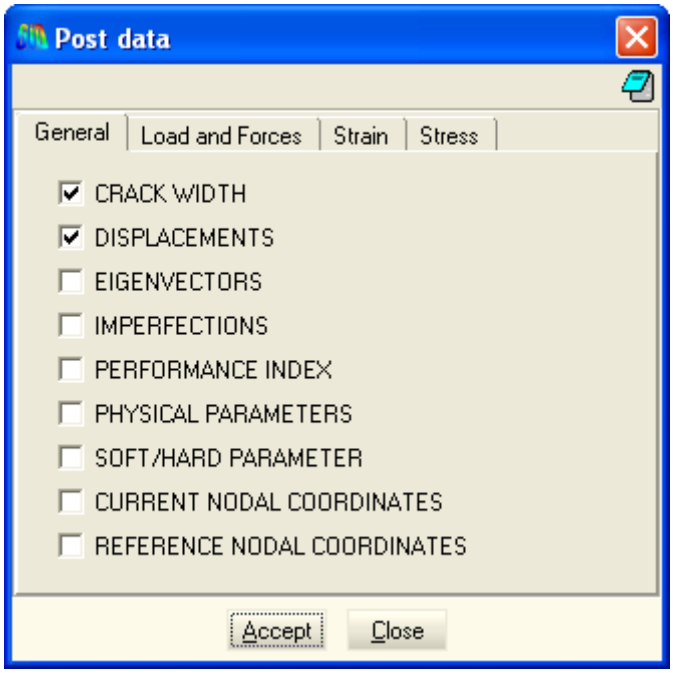


Figure 171: The selection of the data which should be available in the post-process

For example the FRACTURE STRAIN can be chosen. The definition of post data is completed by selecting **`Accept`** button (see Figure 172). Then the button **`Close`** can be pressed and the **GiD** will switch to post-process automatically. But there in the post-process the data from **AtenaWin** has to be imported again.

It is done by the clicking on the AtenaWin icon . Then the FRACTURE STRAIN can be found in the options for the post processing (see Figure 173, to obtain this figure the 35th step has to be selected again).

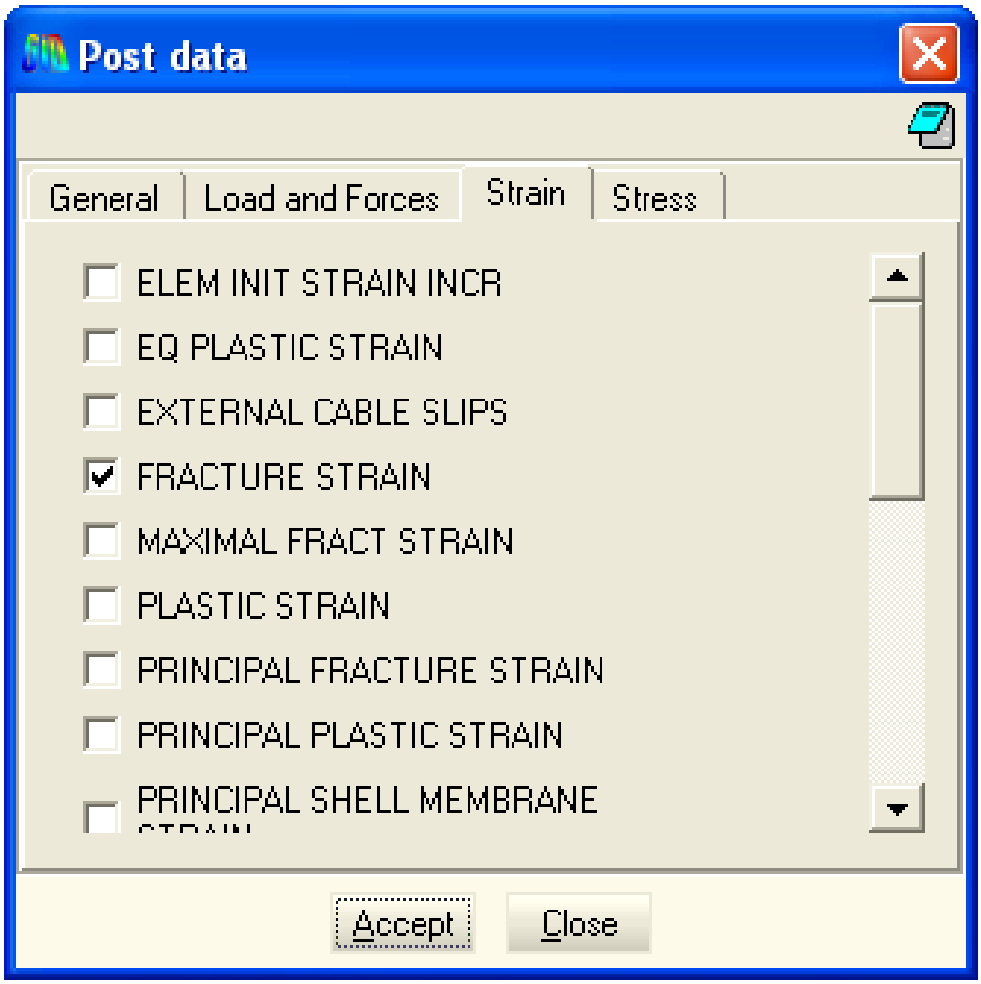


Figure 172: The selection of the FRACTURE STRAIN

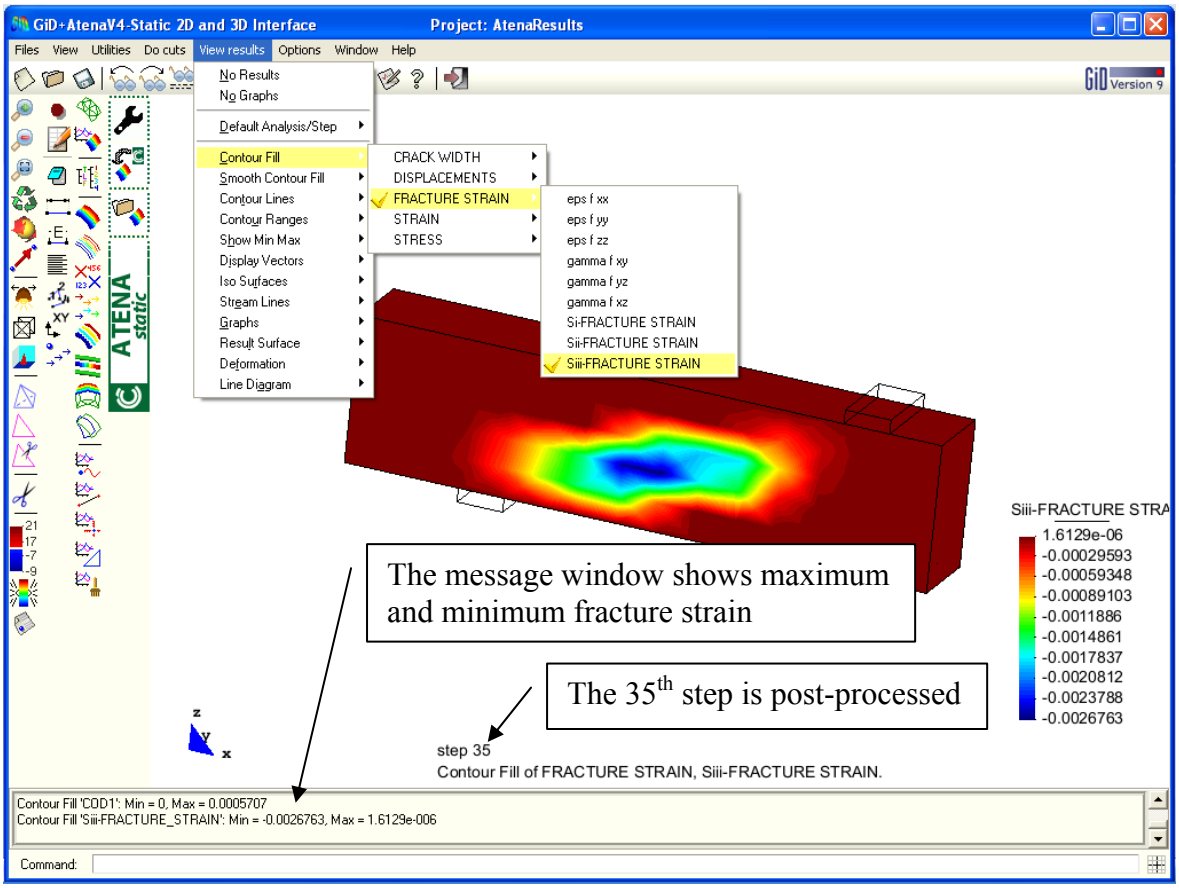


Figure 173: The displayed FRACTURE STRAIN

More post-processing capabilities can be found in the Help of the **GiD** or in the GiD manual [5].

5.2 AtenaWin post-processing

Results can be post-processed also in **AtenaWin**. The L-D graph and Crack width, which have been explained in the chapter 4 (section 4.3 and 4.4) are the few of the many possibilities of post-process in **AtenaWin**.

For post-processing in **AtenaWin** it is important to know how to open results in **AtenaWin**. First of all the **AtenaWin** should be started from the Start menu on your computer (see Figure 174).

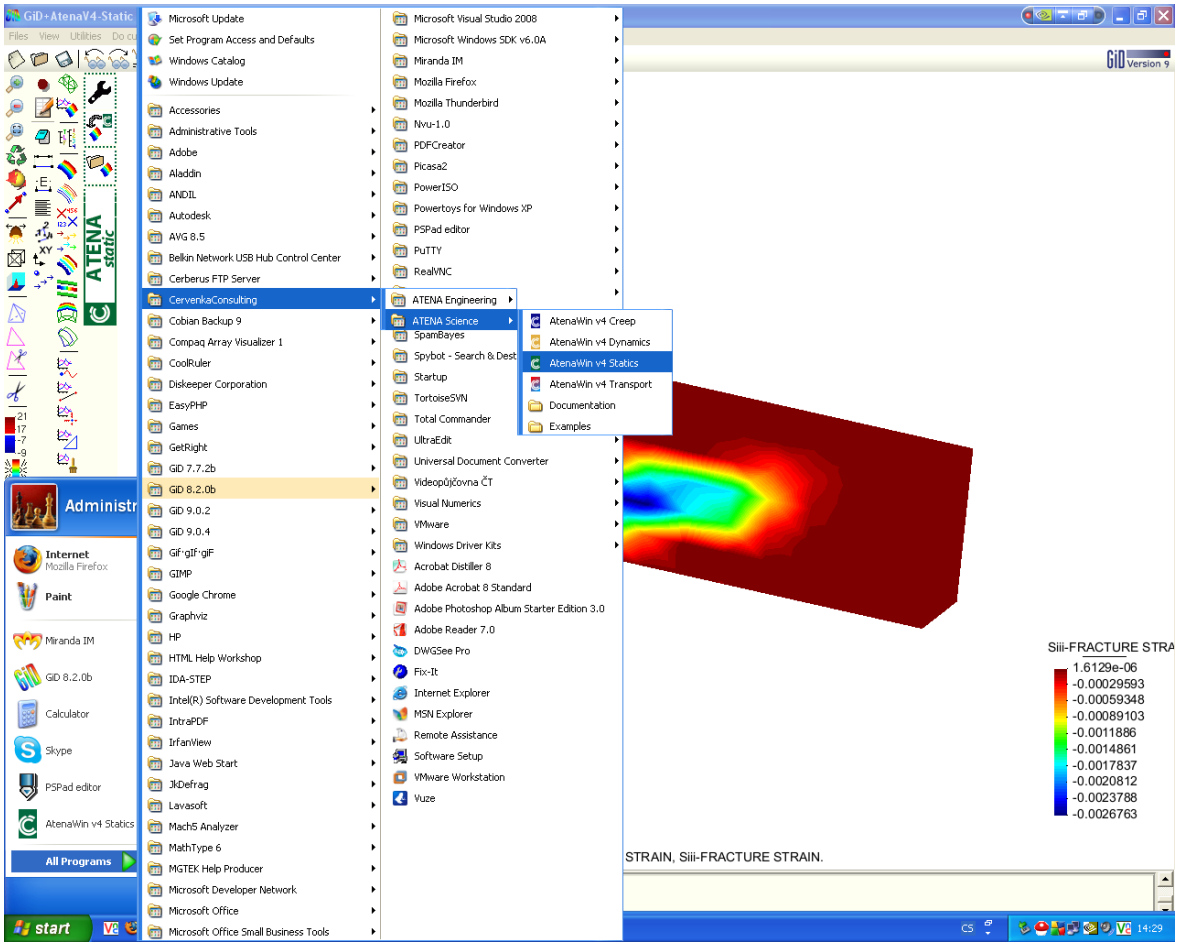


Figure 174: Starting of the AtenaWin

Then results data can be restored by **`Application | Restore FE Model From`** command in the main menu (see Figure 175). The step data file name should be **"3DBeam.00xx"**, where **3DBeam** is task name and **00xx** represents the load step number which should be post-processed. In this case for example the 25th step can be chosen (see Figure 176). Then the display crack width can be defined (see Figure 177). The process of displaying of the crack width is described in the chapter 4.4.

More information about postprocessing can be found in AtenaWin Manual [7].

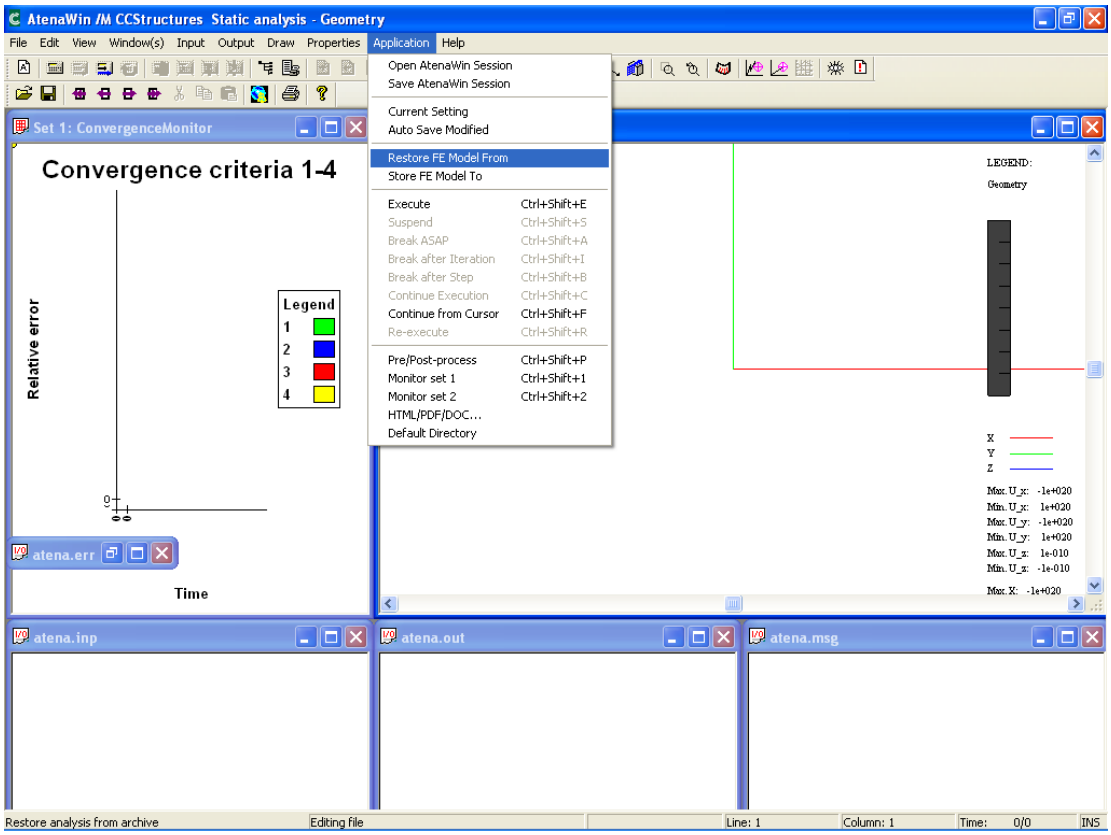


Figure 175: The Restore FE Model From command

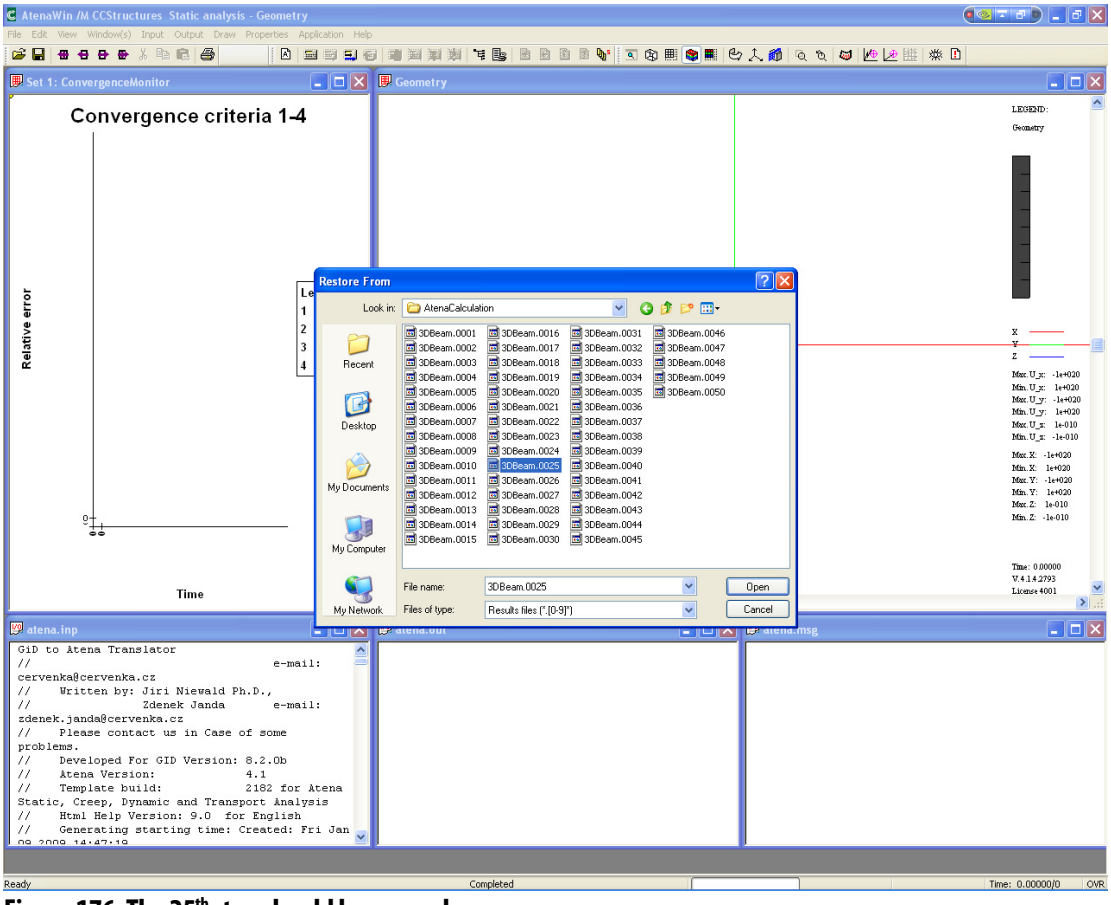


Figure 176: The 25th step should be opened

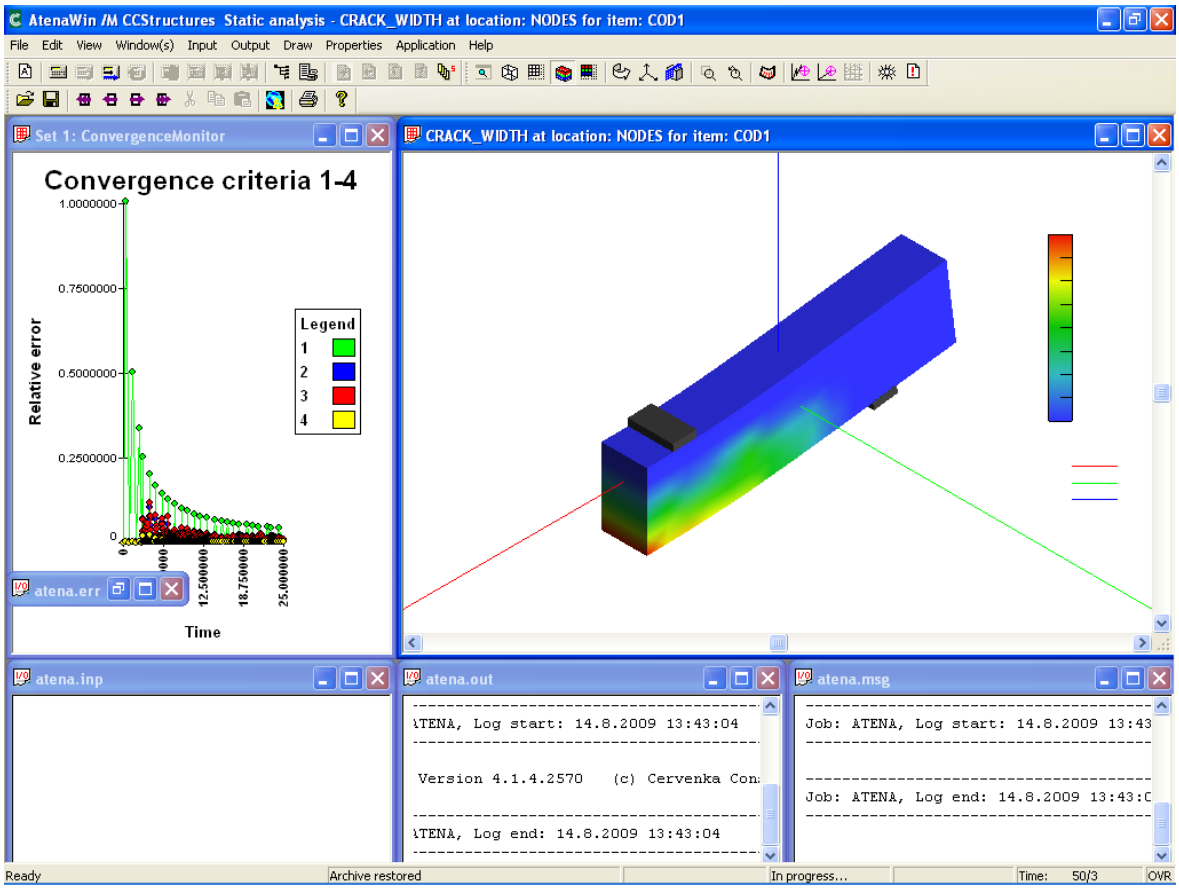


Figure 177: The crack width display of the 25th step

6. CONCLUSION

This tutorial provided a step by step introduction to the usage of **ATENA-GiD** on an example of a reinforced concrete beam without shear reinforcement. Although this example is relatively simple from geometrical and topological point of view, it is not a simple problem from the numerical point of view. Due to the missing shear reinforcement the beam fails by a diagonal shear crack, which is very difficult to capture using smeared crack approach.

This example demonstrates the powerful simulation capabilities of **ATENA-GiD** for modelling the brittle failure of concrete structures. Even with a coarse mesh, which was used in this demonstration example, the diagonal shear crack was successfully captured. Further improvement of the results can be achieved by decreasing the finite element size to for instance 8 elements over the beam height, 4 elements over the beam width and 25 elements over the beam length.

The objective of this tutorial is to provide the user with basic understanding of the program behaviour and usage. For more information the user should consult the user's manual [2]or contact the program distributor or developer. Our team is ready to answer your questions and help you to resolve your problems.

The theoretical derivations and formulations that are used in the program are described in the theory manual [1].

The experienced users can also find useful information in the manual for the analysis module only [4].

7. PROGRAM DISTRIBUTORS AND DEVELOPERS

Program developer: **Cervenka Consulting Ltd.**
Na Hrebenkach 55, 150 00 Prague 5, Czech Republic
phone: +420 220 610 018
fax: +420 220 612 227
www.cervenka.cz
email: cervenka@cervenka.cz

Program distributors:

AUSTRALIA
Palmer Technologies Pty Ltd,
PO Box 1513, Coorparoo DC Q 4151¨
phone: +61 7 38474048
fax: +61-7-339-449-34
e-mail: trish.palmer@palmertechgroup.com.au , www.palmertechgroup.com

FINLAND
A&S Virtual Systems Oy
Laivalahdenkatu 2b, 00880 Helsinky, Finland
phone: +358 207 412 210
fax: +358 9 727 6110
e-mail: juha.airola@virtualsystems.fi, www.virtualsystems.fi

GERMANY
WOELFEL Beratende Ingenieure GmbH u. Co.
Bereich Technische Programme, Max-Planck Strasse 15, D-97204 Hoechberg
phone: +49 931 49708-31
fax: +49 931 49708-15
e-mail: wtp@woelfel.de, www.woelfel.de

INDIA

Computer Solutions

"Shri Sai Kripa", 56, Motilal Nehru Road, Ground Floor,Kolkata - 700 029. West Bengal, India

Mr. Amal Kumar Paul

phone: +91 33 2454 0005

fax: +91 33 2475 5420

email: info@compsolas.com, www.compsolas.com

JAPAN

RESEARCH CENTER of COMPUTATIONAL MECHANICS, Inc. (RCCM)

Togoshi Nl-Bldg. 1-7-1 Togoshi Shinagawa-ku, Tokyo 142-0041, JAPAN

phone: +813 3785 3033

fax: +813 3785 6066

e-mail: junko@rccm.co.jp, www.rccm.co.jp

SOUTH KOREA

CNG SOFTEK

#302, Cheongsan Bldg., 214-6, Poi-Dong, Gangnam-Gu, Seoul, Korea. 135-963

phone: +82-2-529-0841

fax: +82-2-529-0846

e-mail: leeih@cngst.com / cngst@paran.com, www.cngst.com

USA

Ensoft, Inc.

3003 West Howard Lane, Austin, Texas 78728

phone: +1 (512) 244-6464

ext. #201: Sales and Order Status

ext. #208: Technical Support

ext. #202: Consulting Engineering

fax: +1 (512) 244-6067

email: ensoft.ensoftinc@com, www.ensoftinc.com

8. LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2009

[2] ATENA Program Documentation, Part 8, User's Manual for ATENA-GiD Interface, CERVENKA CONSULTING, 2009

[3] ATENA Program Documentation, Part 3, ATENA Examples of Application, CERVENKA CONSULTING, 2005

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2009

[5] GiD Reference Manual, version 9.0.4, International Center For Numerical Methods In Engineering (CIMNE), 2009

[6] Leonhardt and Walther, Schubversuche an einfeldringen Stahlbetonbalken mit und Ohne Schubbewehrung, Deutscher Ausschuss fuer Stahlbeton, Heft 51, Berlin 1962, Ernst&Sohn.

[7] ATENA Program Documentation, Part 7, AtenaWin Description, CERVENKA CONSULTING, 2009